

ME 24-688 – Week 6

Project 1 – Drawing View Creation

Basic View Creation

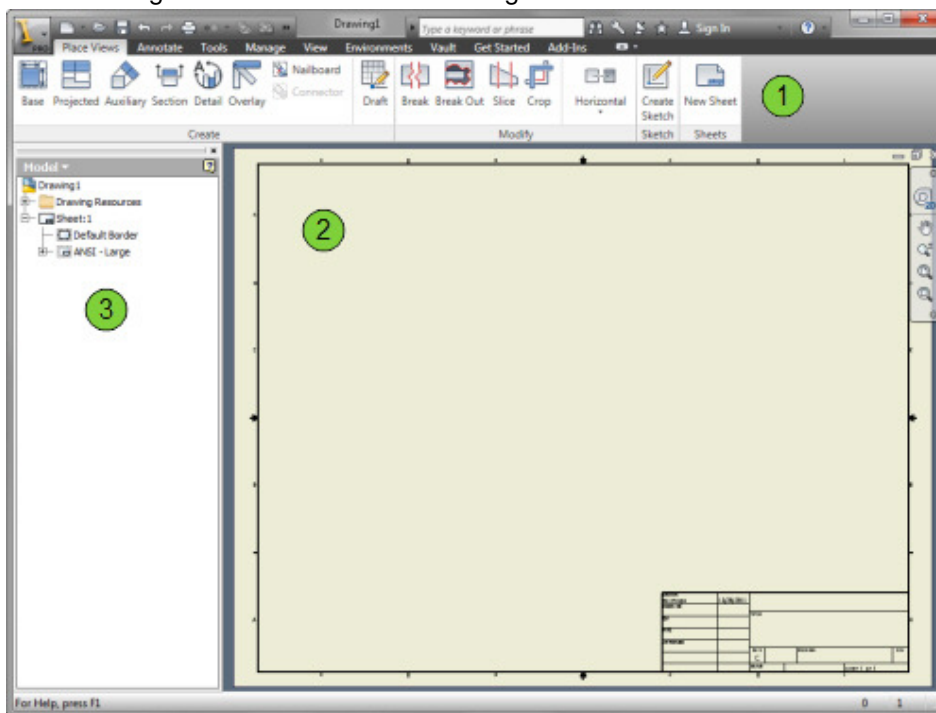
A drawing that can be used to produce a part or assembly as specified is referred to as a production ready drawing. When creating a production-ready drawing, you work in the drawing environment. Within this environment you create the drawing views of the parts or assemblies and add annotations to fully communicate your design requirements and intent.

1. About the Drawing Creation Environment

Overview

The drawing creation environment enables you to create production-ready drawings by creating the necessary views, annotations, notes, and other information needed to produce a part or assembly. The drawing creation environment has three main areas that you use in the creation of a drawing: Ribbon, Drawing Sheet, and the Browser.

The following illustration shows the drawing creation environment



1 Ribbon: The ribbon contains the tools that you use to create views and annotations and the standard tools.

2 Drawing Sheet: The primary, and typically largest, area of the drawing environment is the drawing sheet. The drawing sheet represents the paper on which the drawing is created.

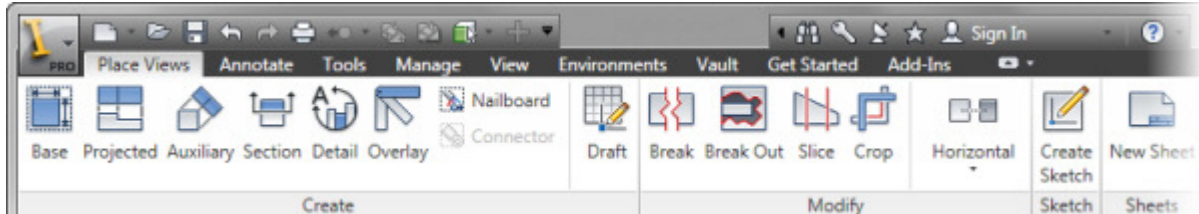
3 Browser: The browser tracks the history of the drawing file and has access to drawing resources such as title blocks, borders, and sheet sizes.

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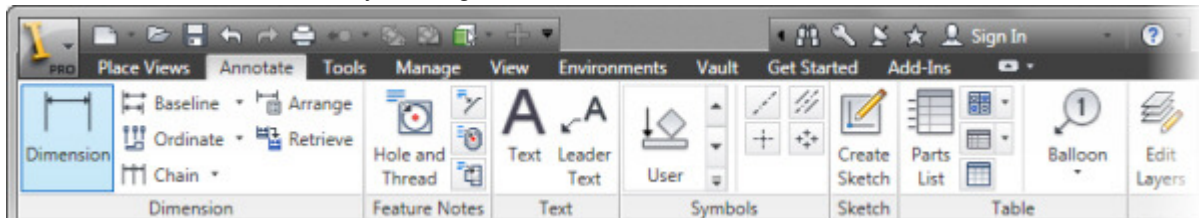
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Drawing Tabs

In the drawing environment, on the ribbon, two tabs are available for creating production-ready drawings. You use the **Place Views** tab to create the various drawing views required to document your parts and assemblies.



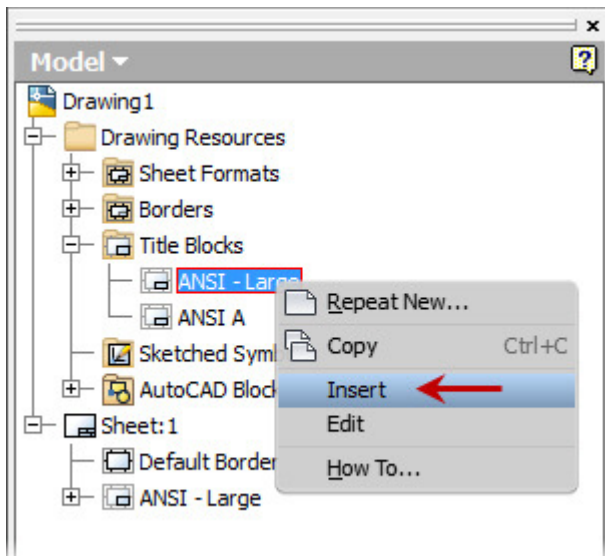
You use the **Annotate** tab to add dimensions, notes, and symbol annotations to the drawing views. You can switch between the tabs by clicking the tab name on the ribbon.



Drawing Environment Browser

In the drawing environment the browser displays the **Drawing Resources** folder, which contains **Sheet Formats**, **Borders**, **Title blocks**, and **Sketched Symbols & AutoCAD Blocks** (The **AutoCAD Blocks** folder is only available when using an Inventor DWG template or drawing file. It also displays each sheet in the drawing, along with the views that you create for each.

In the illustration the cursor is moved to the browser and a new title block is being inserted



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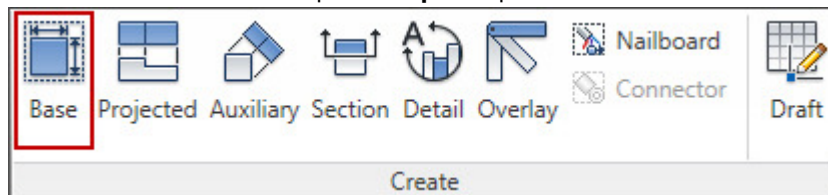
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2. Creating Base Views

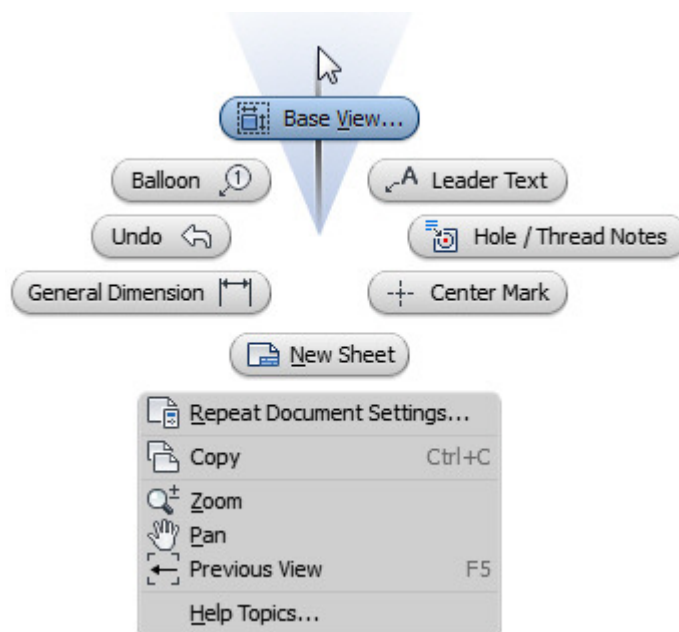
You create a base view to begin creating orthographic views. The base view establishes the original view orientation and scale upon which projected views are based. When you create the base view, you specify the file to be used for the view, the view orientation, scale, and style. After you specify this information, the view is placed onto the sheet and an associative link between the drawing and the part, assembly, or presentation file is established. If the part geometry changes, those changes are reflected in the drawing

Access

Ribbon: **Place Views** tab | **Create** panel | **Base**



Marking Menu: **Base View**

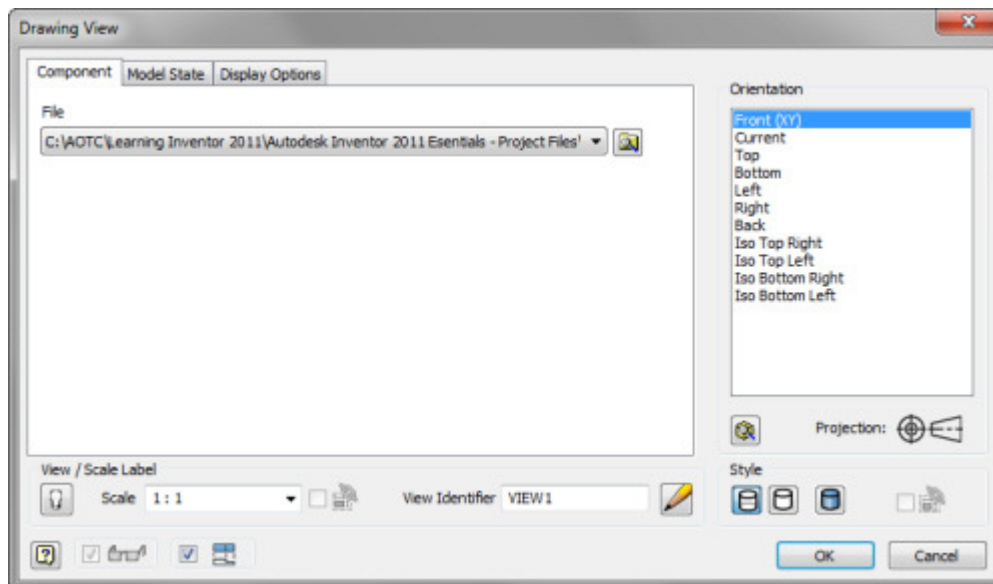


Drawing View Dialog Box

The following options are available in the **Drawing View** dialog box.


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File: Determines the part or assembly file to create its view. If you have a part, assembly, or presentation file open, it is the default file listed. If multiple files are open, you select them from the drop-down list.

Orientation: Determines orientation for the base view. Move your cursor away from the dialog box to see a preview of the view before it is created. The standard view orientations are based upon the origin planes of the file you select.

Change View Orientation:  Opens the model's 3D viewing window. You use standard view tools to define a custom view orientation.

View / Scale Label: Enables you to toggle the display of the view and scale label, select a preset scale value, or enter a custom value for the view. Additionally, you can enter a label for the view or accept the default view label.

- **Scale from Base:** Not available when you create a base view. You use it when you edit projected views
- **Visible:** Displays the scale and view label on the sheet under the view.
- **Edit View Label:** Displays the Format Text dialog box.

Style: Rendering style for the view.

- **Hidden Line:** Hidden lines are displayed.
- **Hidden Line Removed:** Hidden lines are removed.
- **Shaded:** View is shaded using the same colors used in the assembly or part file.

3. Creating Projected Views

The **Projected View** tool enables you to create projected views from any existing view on the sheet. If you select the **Projected View** tool you must select a parent view, then position each projected view. All view positions are previewed in the graphics window prior to the views being created.

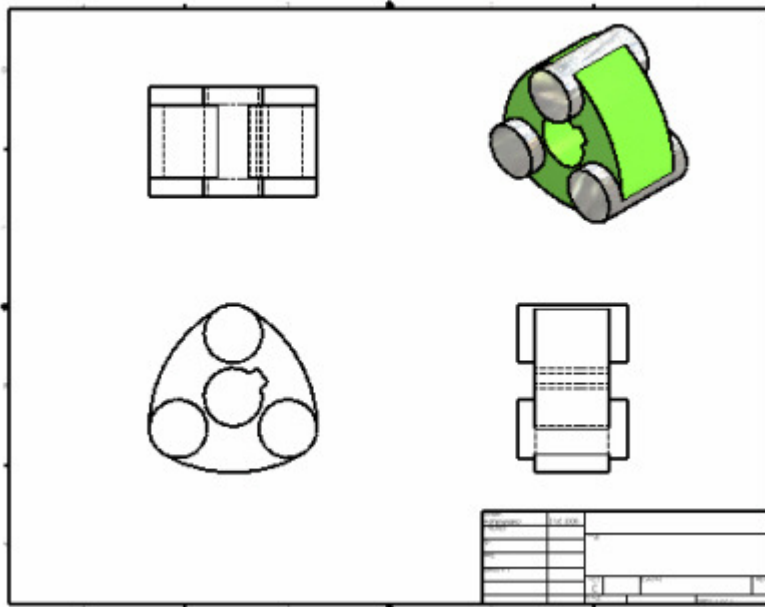
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When you create projected views, the view orientation is automatically determined based on its position on the sheet relative to the base view. If you place the projected view to the right of the base view, it generates a right-side projection of the parent view. If you place the projected view at an angle to the parent view, it generates an isometric view based on the relative position to the parent view. By default, the following view properties are carried over from the base view:

- Scale
- Style (Orthographic Only)

In the illustration, the right, top, and isometric views are projected from the lower left base view

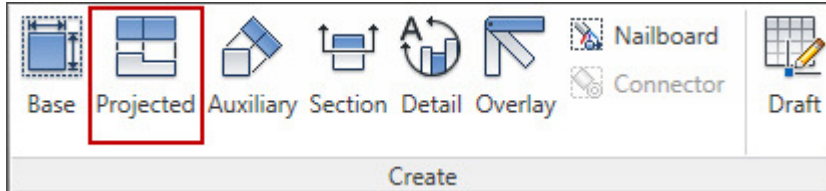


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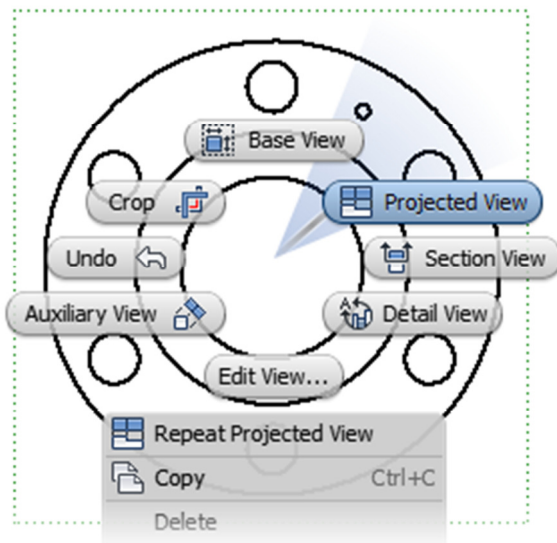
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Access

Ribbon: **Place Views** tab | **Create** panel | **Projected**



Marking Menu: **Base View**



Project Views With Base View Command



When placing a Base View with the Base view tool you can also create projected views immediately after you create a base view without exiting the command. Make sure the Create projected views immediately after base view creation box on the **Drawing View** dialog box is checked. Click to place the required projected views

Drafting Standards Projection Setting

By default projected view are created using the Third Angle project method. This setting can be changed in the **Drafting Standards** dialog box. The First Angle projection method is also available. Style and Standards editing is covered in the **Drawing Standards and Resources** unit.

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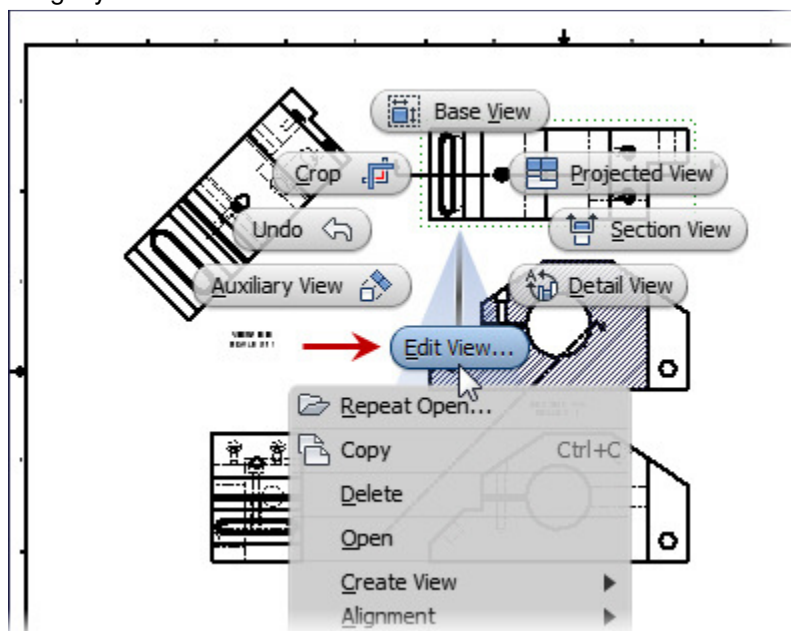
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4. Properties of Editing Views

Overview

After you create base and projected views, you can edit the view properties using the **Drawing View** dialog box. Depending on the type of view, base or projected, different options are available for editing.

When you edit a base view, you can change the scale and style properties. However, while editing a projected view, you can change these properties only if you clear the **Scale from Base** or **Style from Base** options. In a projected view, these properties are linked to the base view to ensure the same scale and the same rendering style across views.

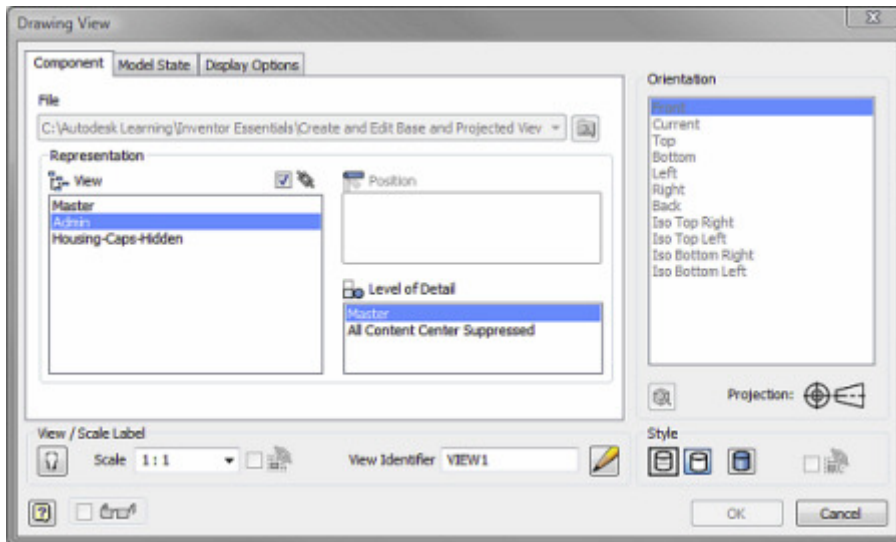


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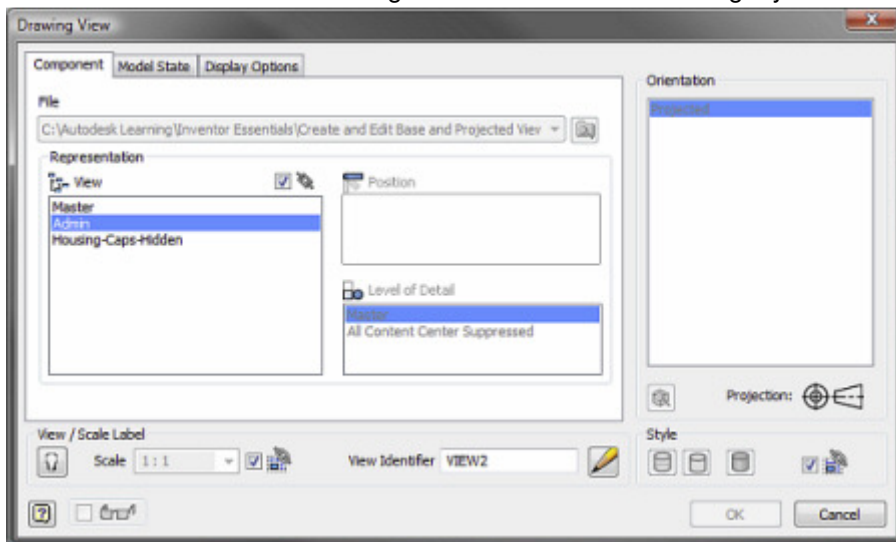
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Editing a View

When you edit a base view, you can edit any option that is not grayed out. If you change the scale factor on the base view, all projected views with the **Scale from Base** option selected are updated to reflect the new scale factor.



When you edit a projected view, you can edit any option that is not grayed out. Clear the **Scale** and **Style from Base** check boxes to change the view scale or rendering style.

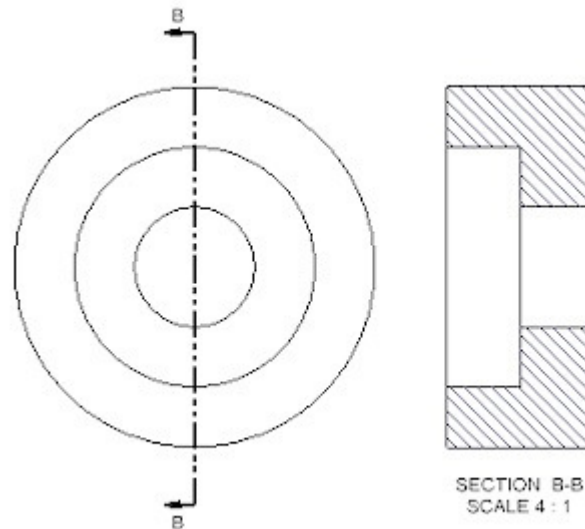


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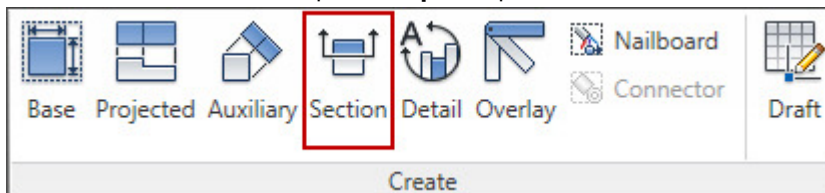
5. Creating Section Views

In order to create a section view, you must have at least one view on the sheet on which the section line is drawn. After drawing the section line, you choose a side of the current view for the section view. The section view is generated based on the direction of sight relative to the view being sectioned.

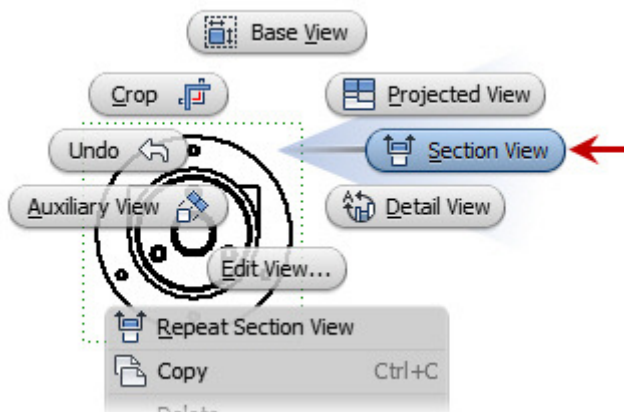


Access

Ribbon: **Place Views** tab | **Create** panel | **Section**



Marking Menu: **Section View**

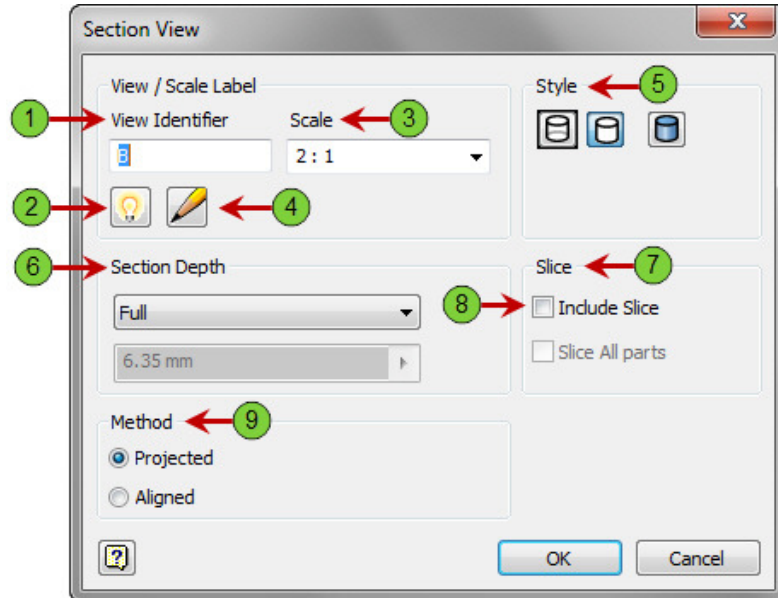


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Section View Dialog Box

The following options are available in the **Section View** dialog box.



- 1 **View Identifier:** Use to specify a view label or accept the default value.
- 2 **Toggle Label Visibility:** Displays the label and view scale on the sheet.
- 3 **Scale:** Scale factor for the section view.
- 4 **Format Text:** Access the Format Text dialog box.
- 5 **Style:** Rendering style for the view.
 - Hidden Line
 - Hidden Line Removed
 - Shaded
- 6 **Section Depth:** Section depth for the view.
 - Full: Section depth is calculated through the entire part or assembly.
 - Distance: Measured from the section line to calculate the section view. All geometry outside of the calculated distance is ignored and is not displayed in the view.
- 7 **Slice:** Depending on browser settings, when checked, some parts are sliced, and some sectioned.
- 8 **Slice All Parts:** Browser settings are overridden and all parts in the view are sliced according to the section line geometry. Parts not crossed by the section line are not included in the view. Section Depth fields are disabled.
- 9 **Method:** Use the **Projected** method to project the lines orthogonally to the section views position. The **Aligned** method projects section geometry perpendicular to each segment of the section line. This option only appears if the section line contains more than one segment.

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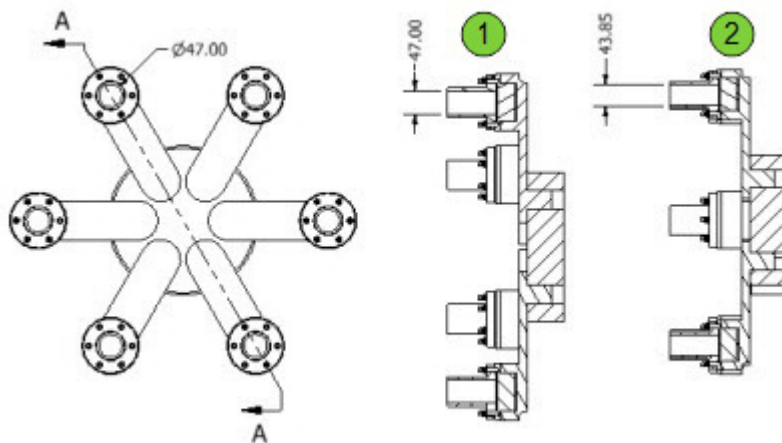
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Section View Projection Methods

The difference between **Aligned** and **Projected** section views is visible when the section line cuts through openings in the view at an angle. If you were to add a dimension to an opening in an **Aligned** section view cut at an angle, the dimension value returned would be identical to a dimension value placed on the same feature in the base view. In a **Projected** view the dimension value returned would be the perpendicular distance from where the section line intersects the opening.

In the illustration, two section views are created using identical section lines. The differences in the

Aligned ② and **Projected** ① **Views** are as follows. The dimensional value of the feature in the **Aligned** view matches the same value on the same feature in the base view. In the **Aligned** view, since you are always viewing the section perpendicular to the section line, you do not see the patterned component in the middle of the view. The projected view is elongated to allow for the increase cross section of the features.



6. Creating Detail Views

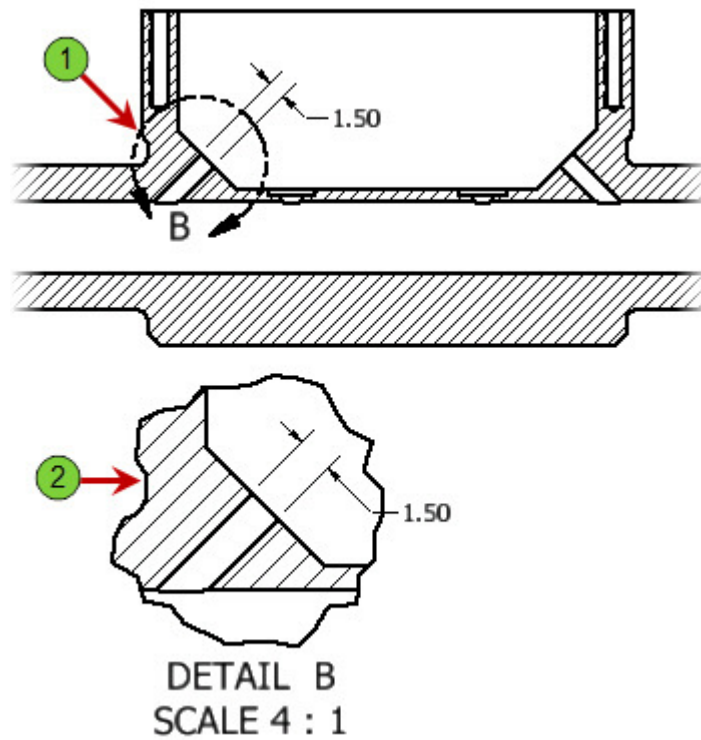
You use the Detail View tool to create detail views of an existing view in the drawing. When you use Detail Views, you define the detailed area by specifying a center point and a rectangular or circular fence. All geometry contained within the detail view rectangle or circle is included in the detail view.

When you create a detail view, you magnify an area of the drawing while creating an associative link between the original view and the detail view. If the geometry being magnified changes in the original view, those changes are reflected in the detail view. Also, the placement and readability of dimensions in these areas of the drawing are simplified.

A detailed view is associated with the main view, and any changes that affect geometry within the main view are reflected in the detail view automatically. Although the view is scaled, as is true of other scaled views, when you place dimensions on geometry within the view, the dimensions reflect the actual geometry size.

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1 Detail view circle

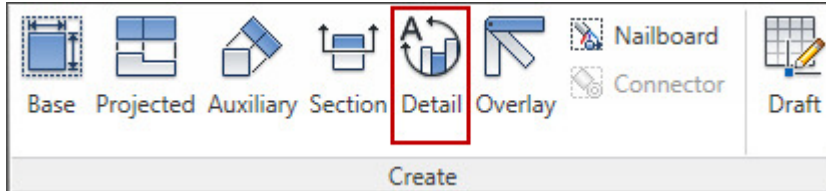
2 Scale detail view with dimensions

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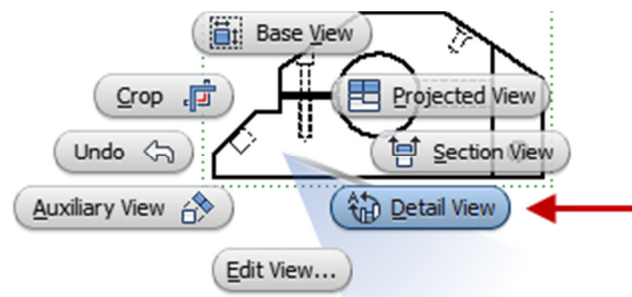
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Access

Ribbon: **Place Views** tab | **Create** panel | **Detail**



Marking Menu: **Detail View**



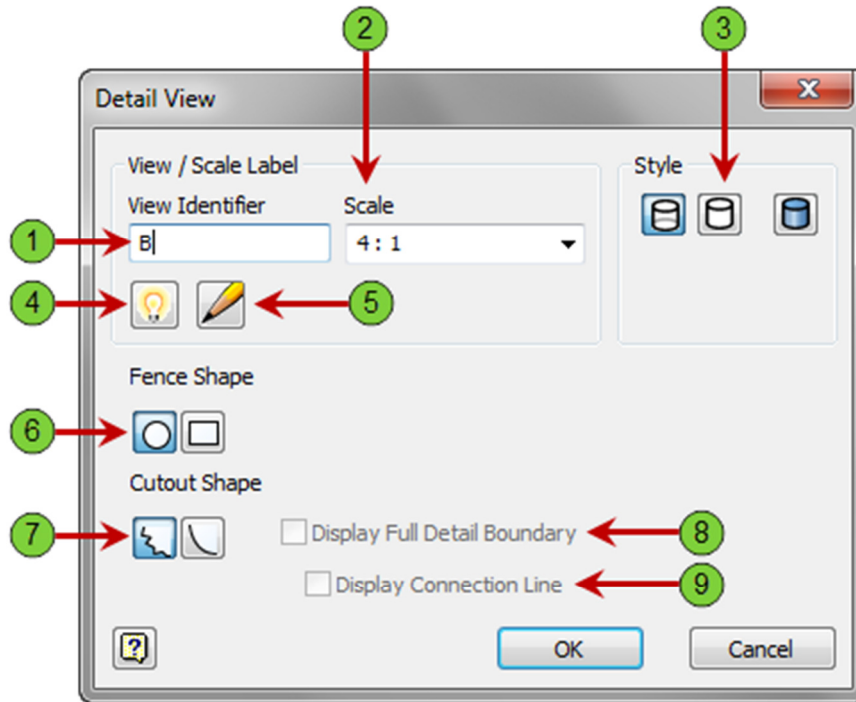
Shortcut Menu: **Create View** | **Detail View**

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Detail View Dialog Box

The following illustration shows the **Detail View** dialog box.



The following options are available in the **Detail View** dialog box.

- 1 **View Identifier:** Use to specify a view label or accept the default value.
- 2 **Scale Use:** to specify the scale factor for the detail view. Select from the list or manually enter a custom value.
- 3 **Style:** Determines a rendering style for the view.
 - Hidden Line
 - Hidden Line Removed
 - Shaded
- 4 **Toggle Label Visibility:** When selected, the view scale label is visible on the sheet.
- 5 **Edit View Label:** Use to access the Format Text dialog box.
- 6 **Fence Shape:** Determines a fence shape for the view.
 - Circular
 - Rectangular
- 7 **Cutout Shape:** Specify the cut line as Jagged or Smooth.

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8 Display Full Detail Boundary: If Smooth cutout shape is selected, select this option to have a boundary drawn around the detail view.

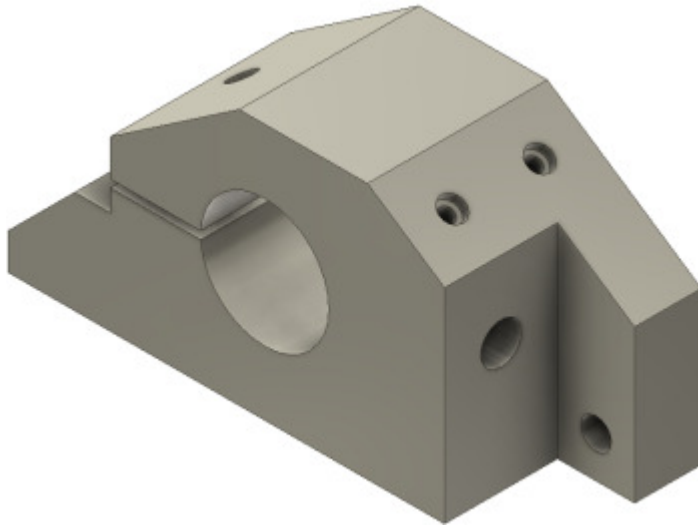
9 Display Connection Line: If the Display Full Detail Boundary option is selected, select this option to have a line drawn between the detail view boundary in the parent view and the boundary around the detail view.

7. Project: Create and Edit Base and Projected Views

In this portion of the project you navigate the drawing creation environment to create projected views and change the sheet size.

Instructions

1. Open *Aux-Clutch-Lever.ipt*.

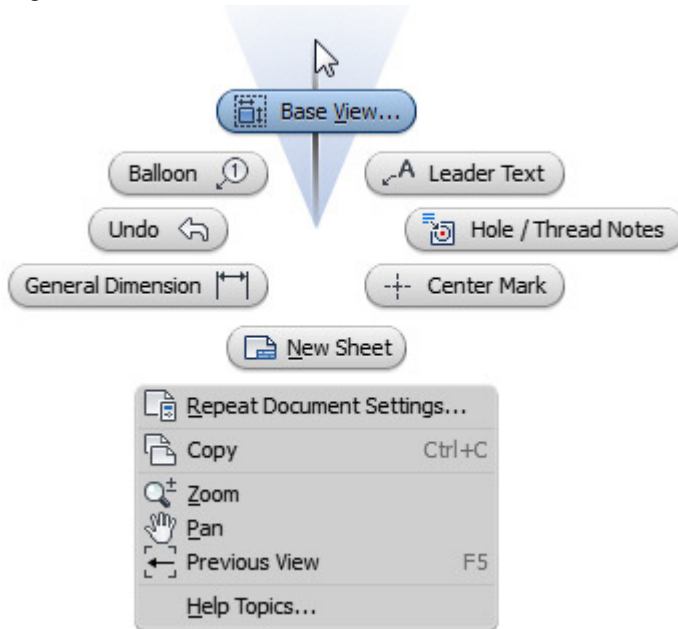


2. Start a new drawing; from the Metric Tab double-click the **ANSI (mm).dwg** template.


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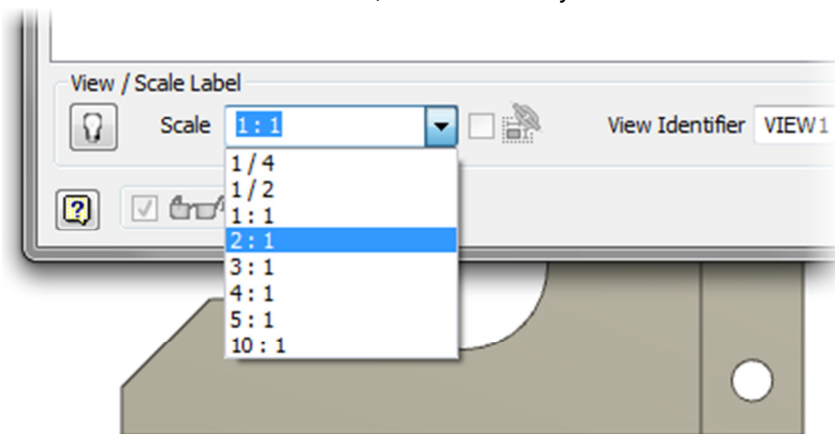
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3. Right click on the sheet and select **Base View**



4. Place a Base View

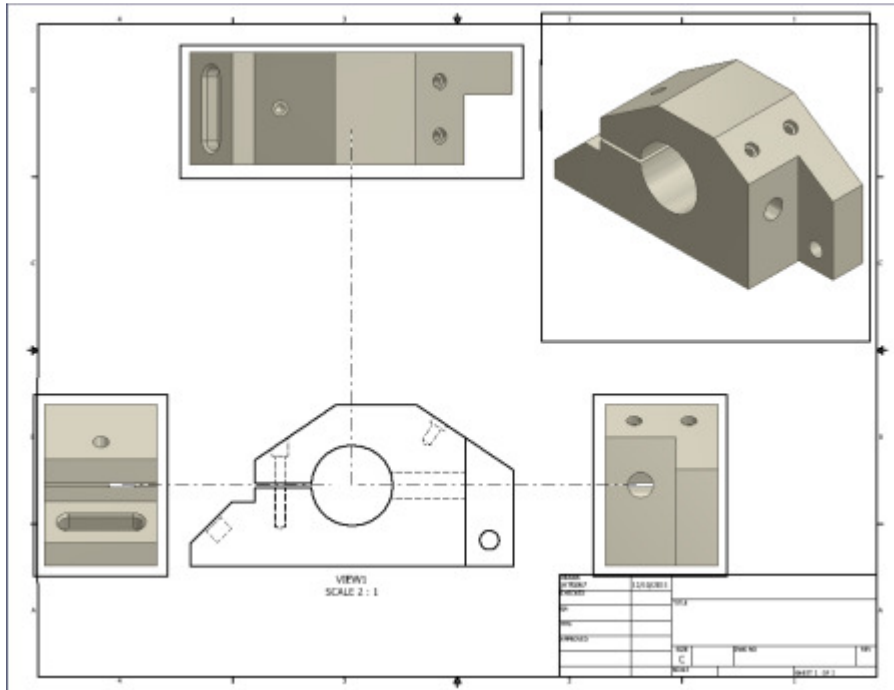
- Under View/Scale label, turn on visibility  and select **2:1**.



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5. Place the project views as shown below

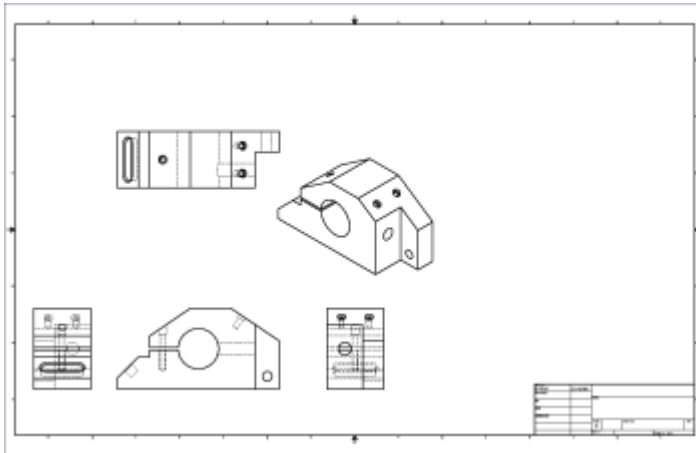


The following link shows the previous two (2) steps: <http://www.screencast.com/t/O2VUaAfJ5>

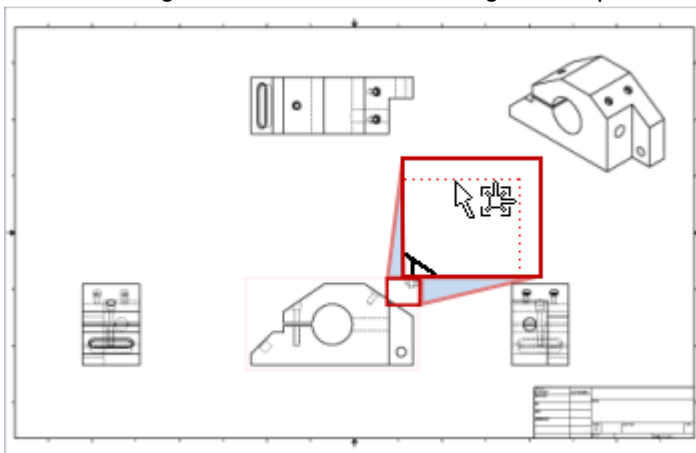
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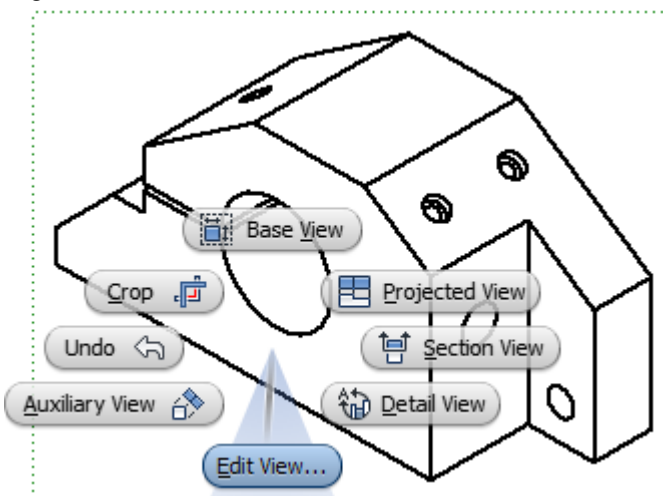
6. Change the sheet size so all the views fit one sheet:
 - In the browser, right-click *Sheet:1*. Click **Edit Sheet**.
 - From the **Size** menu, select **D**. Click **OK**.



7. Click and drag the border of each drawing view to position them as shown.



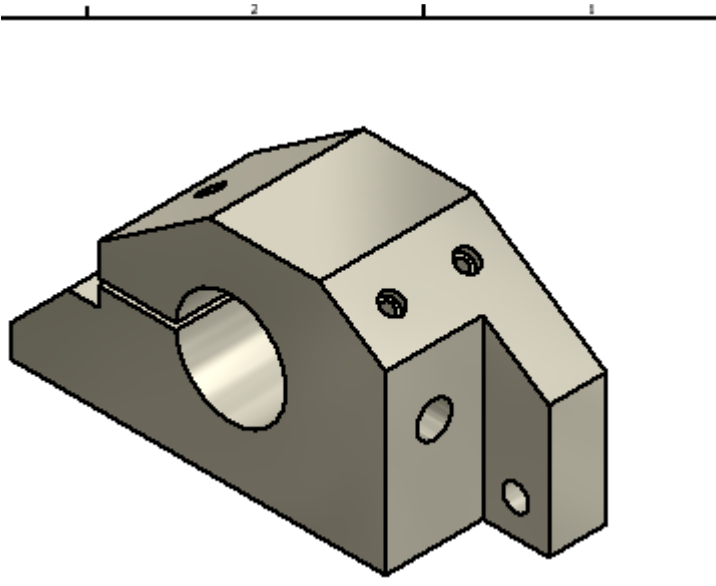
8. Right-Click on the Isometric view and select Edit View from the marking menu.



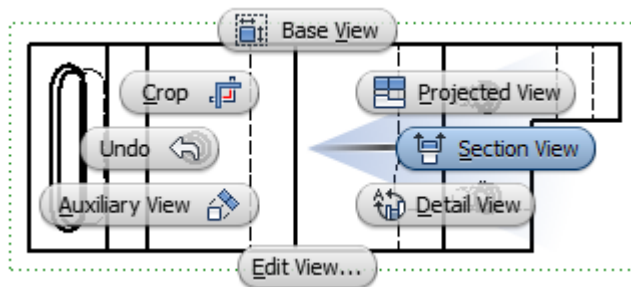
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9. Click the **Shaded Style** button in the **Drawing View** dialog box. Click **OK**.



10. Right-Click on the **Top** projected view and select **Section View** from the marking menu.



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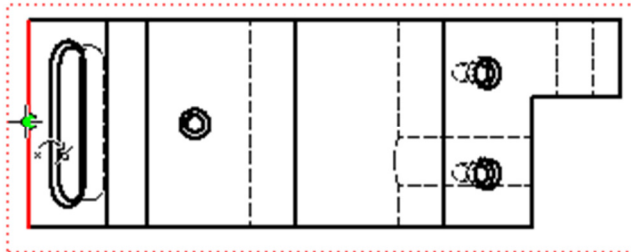
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8. Project: Create and Edit Section Views

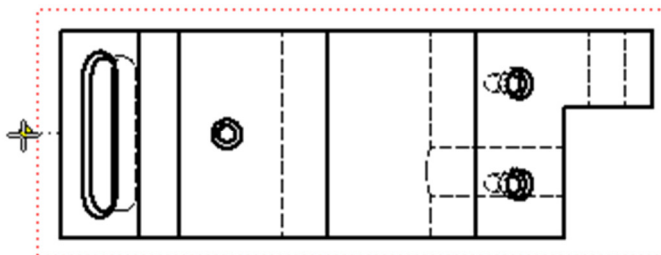
In this portion of the project, you create section views of an assembly. After creating the section view, you turn off sectioning for some components. You edit the section by moving the section line and changing the hatch pattern applied to some components.

Instructions

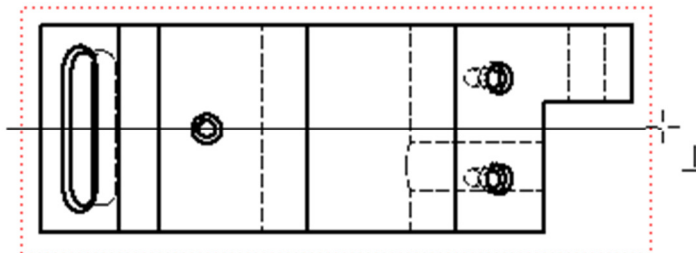
11. Touch midpoint of the left vertical edge with the cursor, but do **NOT** click. A green dot inferring the midpoint will be displayed. Do not select this point.



12. With the midpoint highlighted, move the cursor to the left of the view. You should see the dotted line indicating the point is being projected as shown. Left-click near this point to start the section line.



13. Move the cursor to the right of the view and click. This action constrains the section line perpendicular to the line.



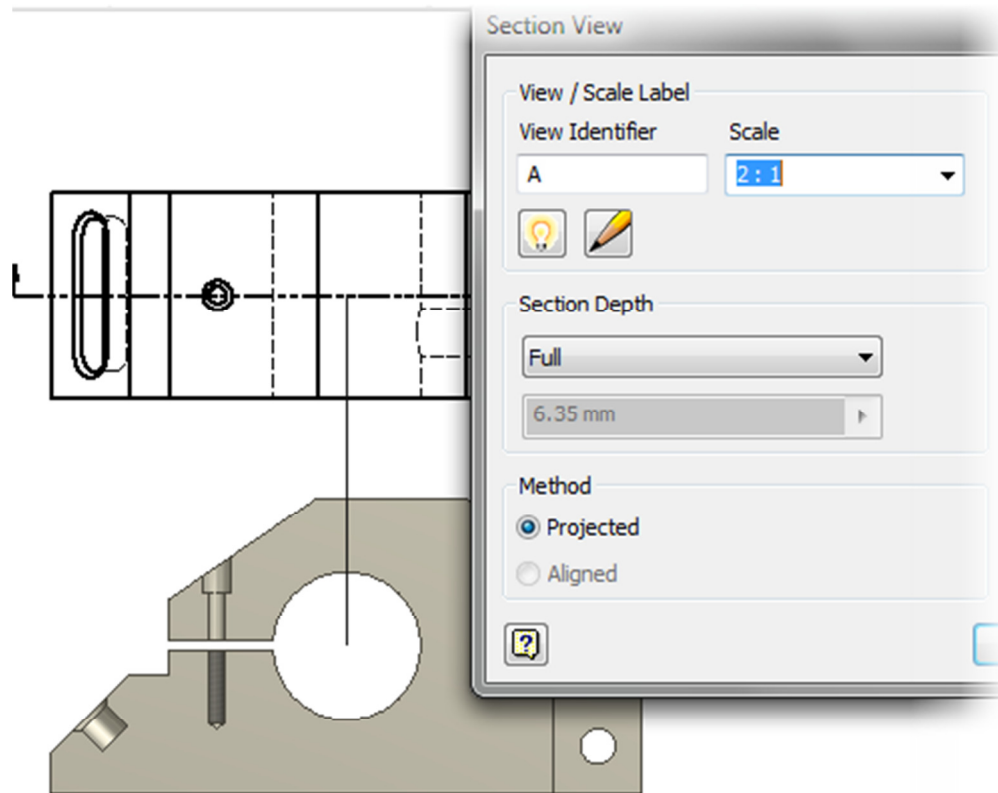
14. Right-click in the graphics window. Click **Continue**.

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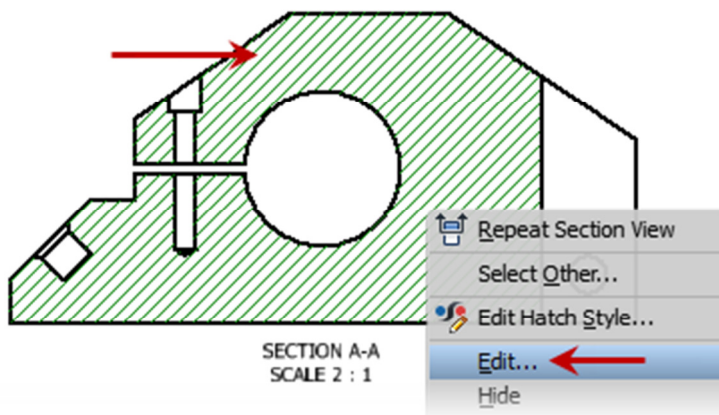
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15. Define the label and create the section view.

- For View Identifier, enter **A**.
- Move the down cursor, and click to position the section view as shown.



16. Right-click in the hatch pattern indicated in the following illustration. Click **Edit**.

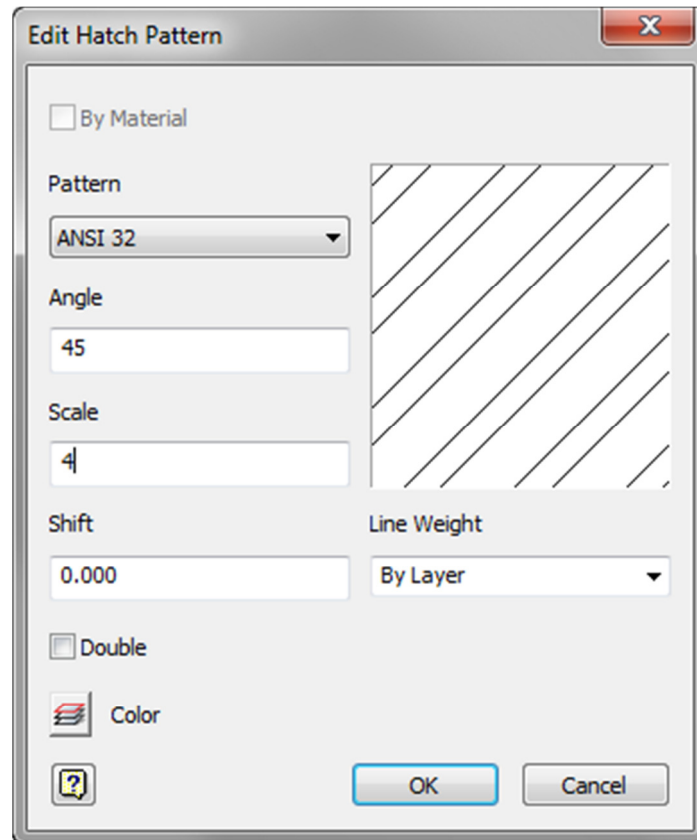


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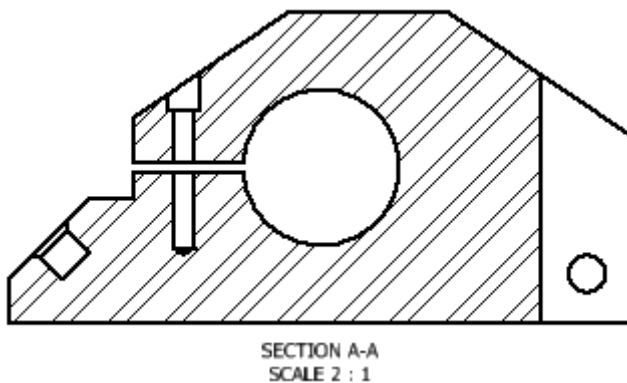
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17. In the **Edit Hatch Pattern** dialog box, under **Pattern**, enter the following values:

- Select **ANSI 32**.
- In the Scale field, enter **4**
- Click **OK**.



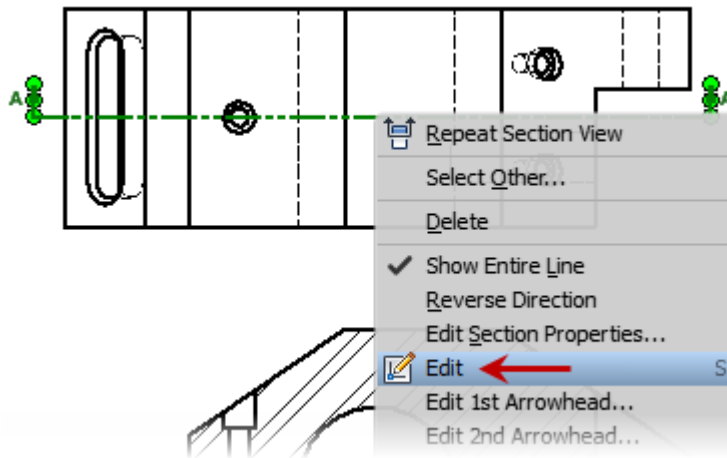
18. Notice the change in appearance in the section view.



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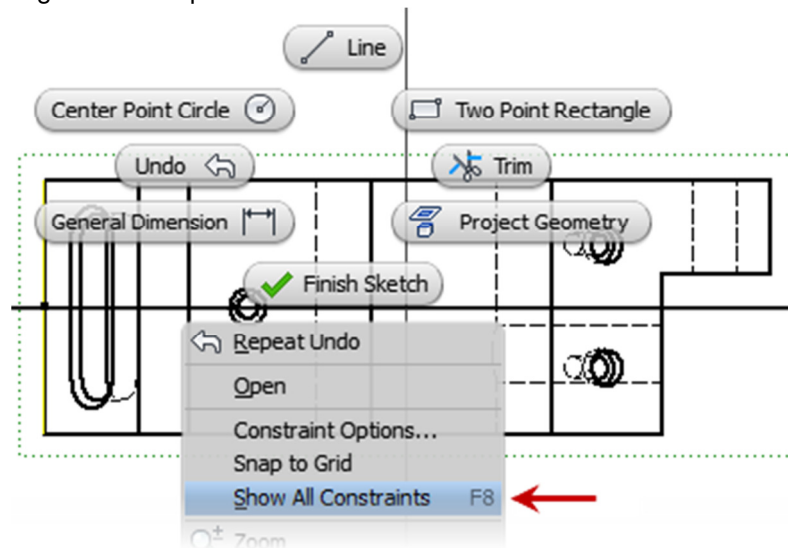
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19. Right-click the section line. Click **Edit**.



20. To show the constraints made during the creation of the section line:

- Right-Click in space and select **Show All Constraints**.

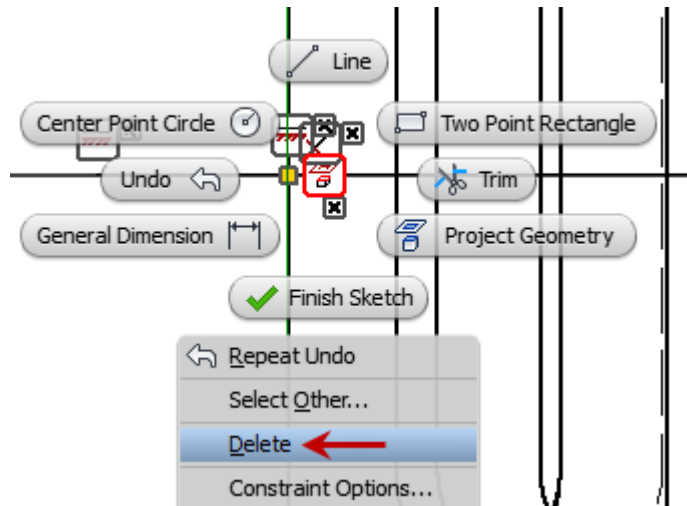


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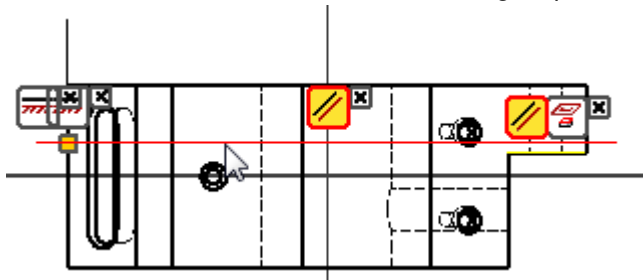
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21. To delete projected constraint to edit the section line position:

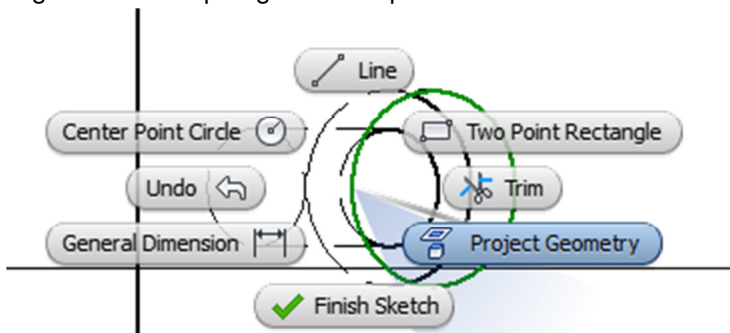
- Right-click on the **Projected Geometry** glyph
- Select **Delete**.



22. You can now click the sketch line and drag it upward. Click to reposition the sketch line.




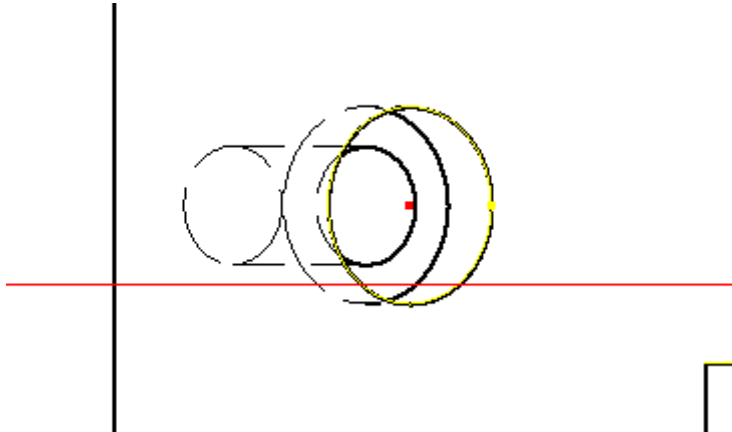
23. Right-click the top edge of the top counter bored hole and select **Project Geometry**



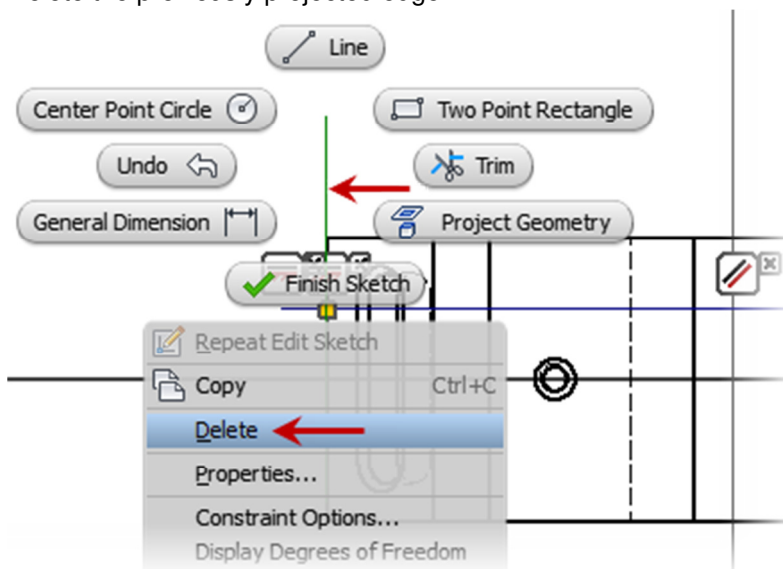
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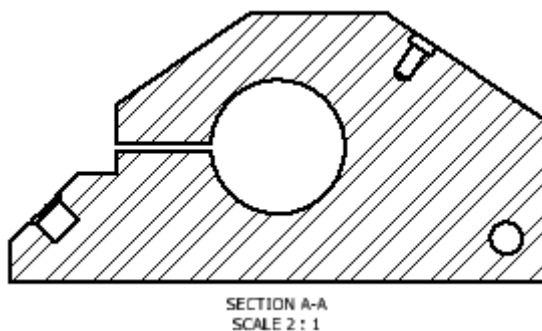
24. Place a **Coincident**  on the line and the center point of the project edge



25. Delete the previously projected edge.



26. Right-click in the graphics window. Click **Finish Sketch**.
Notice the difference in the section view.



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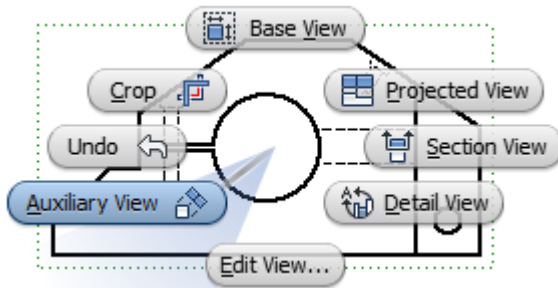
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9. Project: Create and Edit Auxillary Views

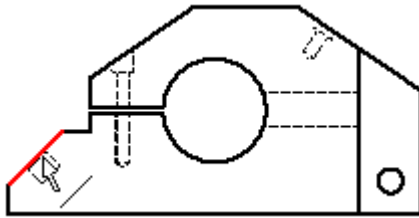
In this portion of the project you navigate the drawing creation environment to create auxillary views and edit view alignment.

Instructions

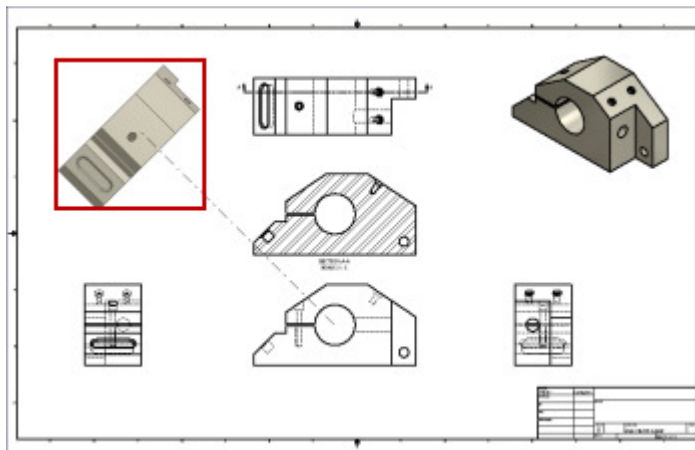
27. Right-Click on the **Base View** and select **Auxillary View** from the marking menu.



28. Select the highlighted edge.



29. Place the view as shown in the illustration.

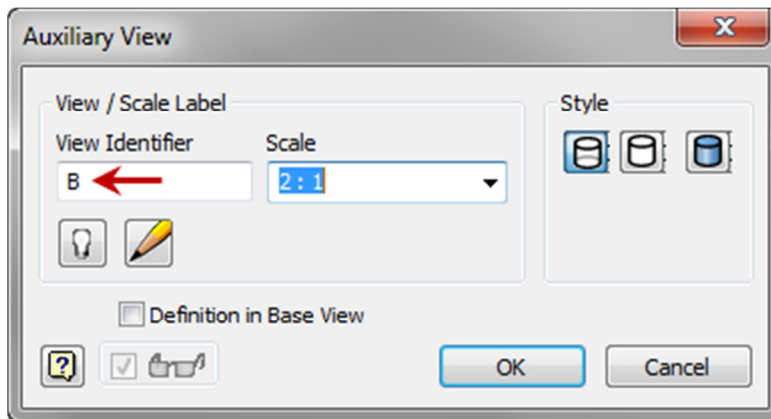
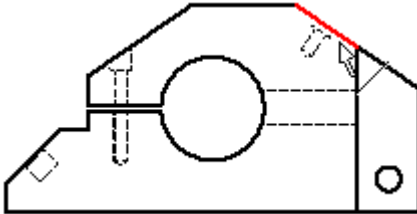


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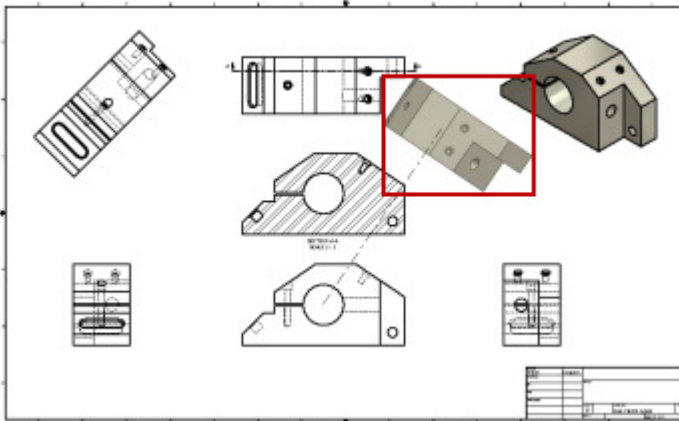
Project 1 – Drawing View Creation

30. Place another **Auxiliary View** using the highlighted edge

- In the **View Identifier** box enter **B**



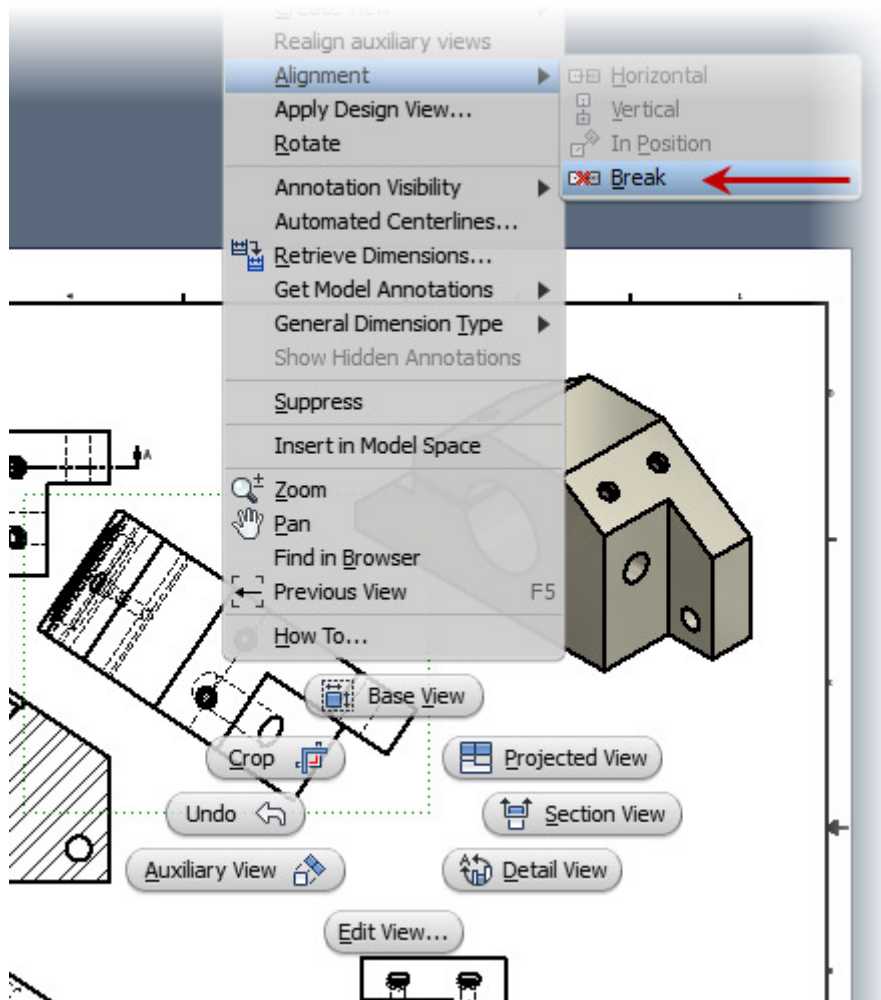
31. Place the view as shown in the illustration.



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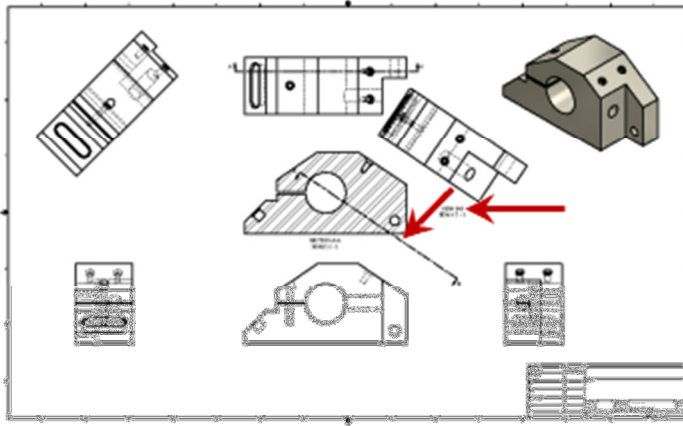
32. Right-Click on the **Auxiliary View** and select **Break** from the **Alignment** flyout.



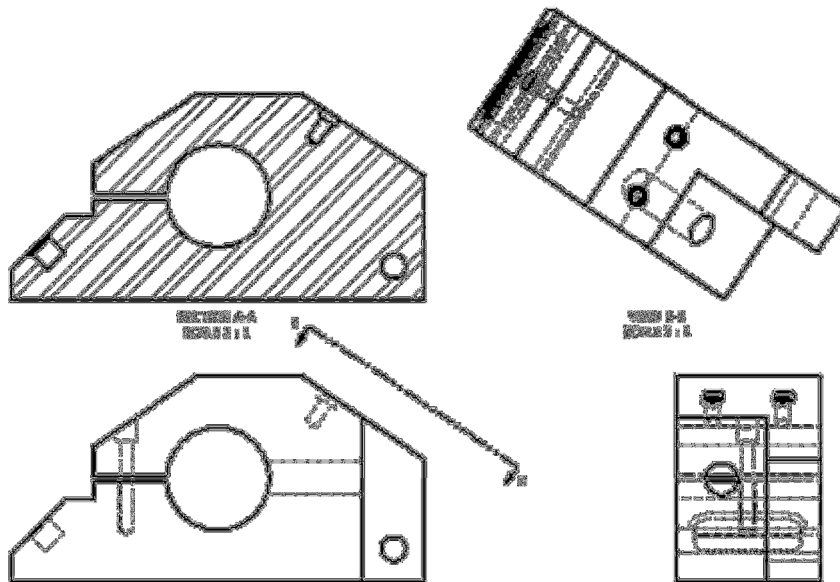
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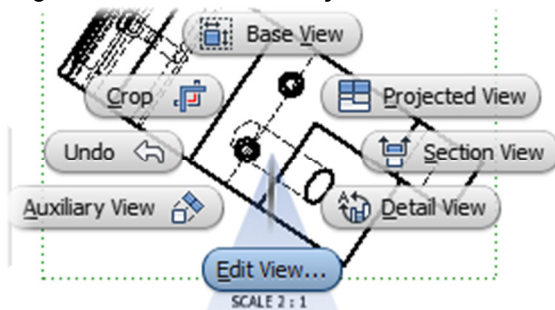
33. Notice the **View Identifier** and the **Definition in Base View** is shown.



34. Reposition the view and resize the definition



35. Right-Click on the **Auxiliary View** and select **Edit View...** from the marking menu.

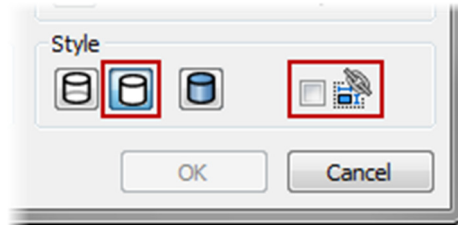


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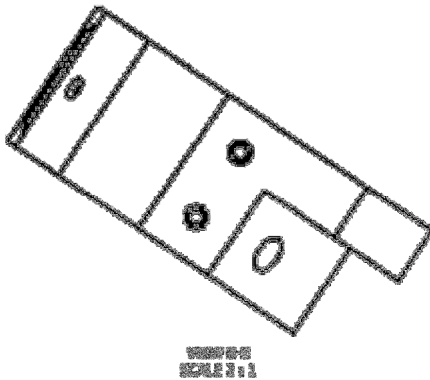
Project 1 – Drawing View Creation

36. Change the display **Style** to **Hidden Lines Removed**:

- Remove the check from the **Style from Base** checkbox.
- Select **Hidden Lines Removed** under the style.



37. Notice the change in appearance in the auxiliary view.



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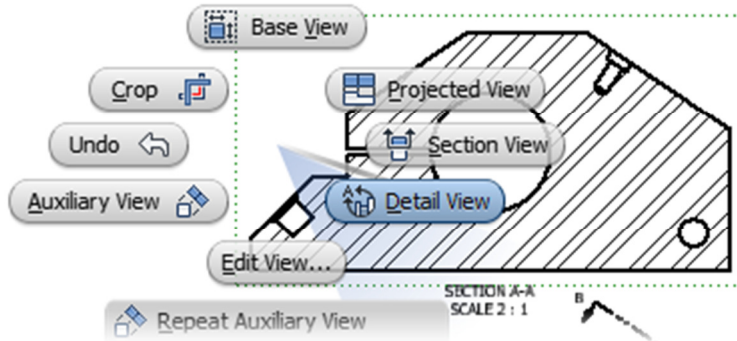
Project 1 – Drawing View Creation

10. Project: Create and Edit Detail Views

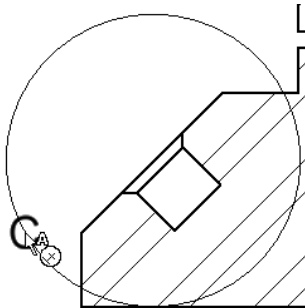
In this portion of the project, you create and edit detail views to magnify critical features of a hydraulic reservoir.

Instructions

38. Right-Click on the **Section View** and select **Detail View** from the marking menu.



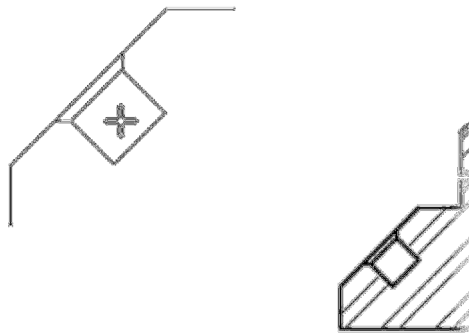
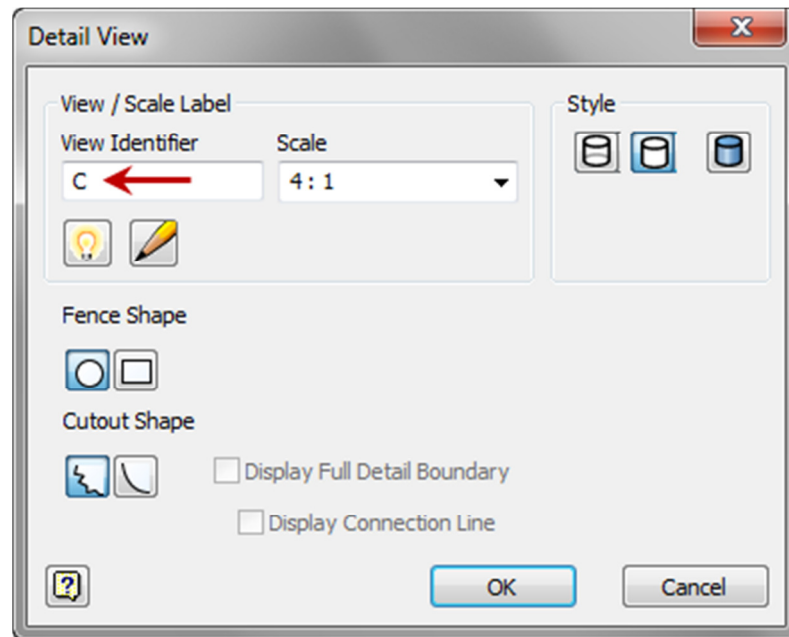
39. Place the Detail circle as shown



40. Place the **Detail View** using the highlighted edge
- In the **View Identifier** box enter **C**

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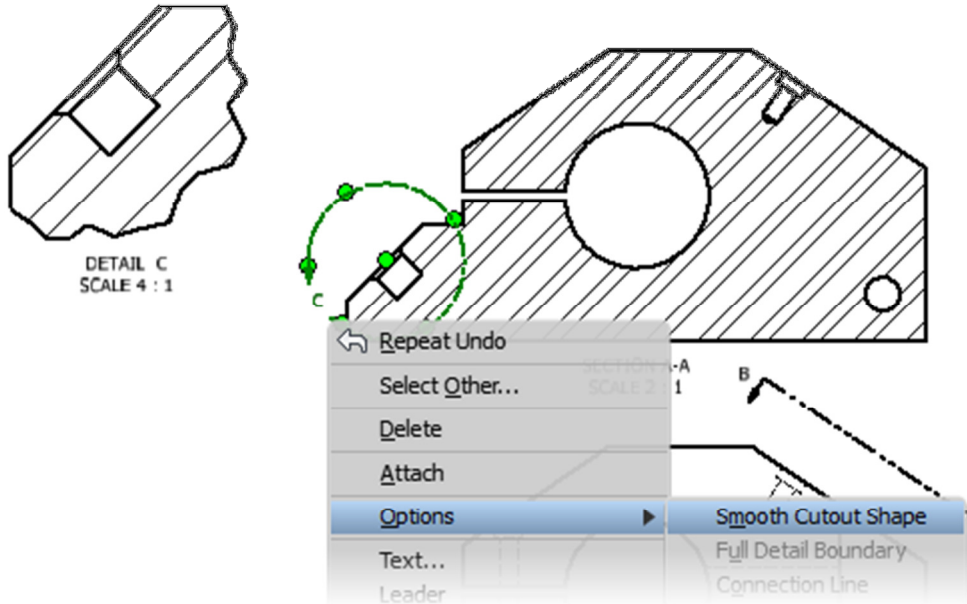
Project 1 – Drawing View Creation



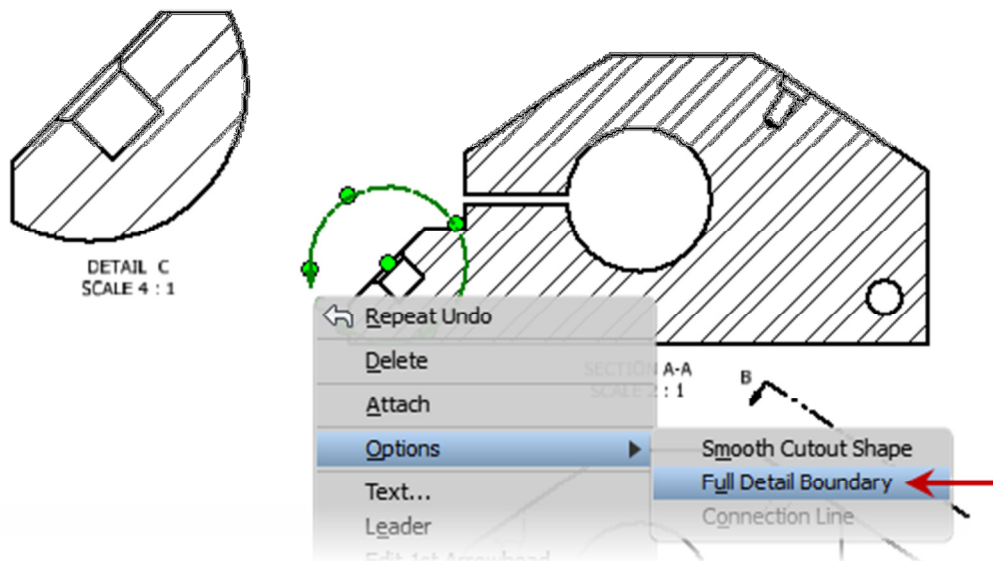
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41. Right-Click on the **Detail View** identifier and select **Smooth Cutout Shape** from the **Options** flyout.



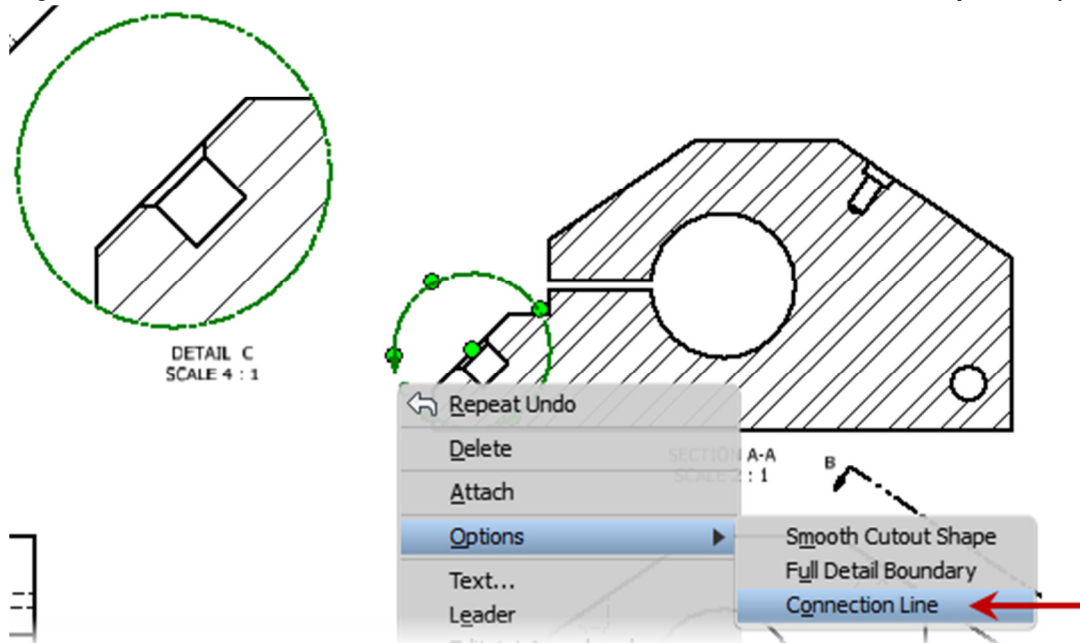
42. Right-Click on the **Detail View** identifier and select **Full Detail Boundary** from the **Options** flyout.



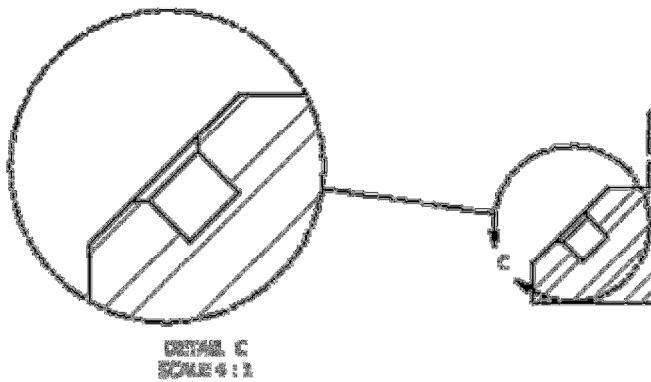
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43. Right-Click on the **Detail View** identifier and select **Connection Line** from the **Options** flyout.



44. Notice the change in appearance in the detail view.



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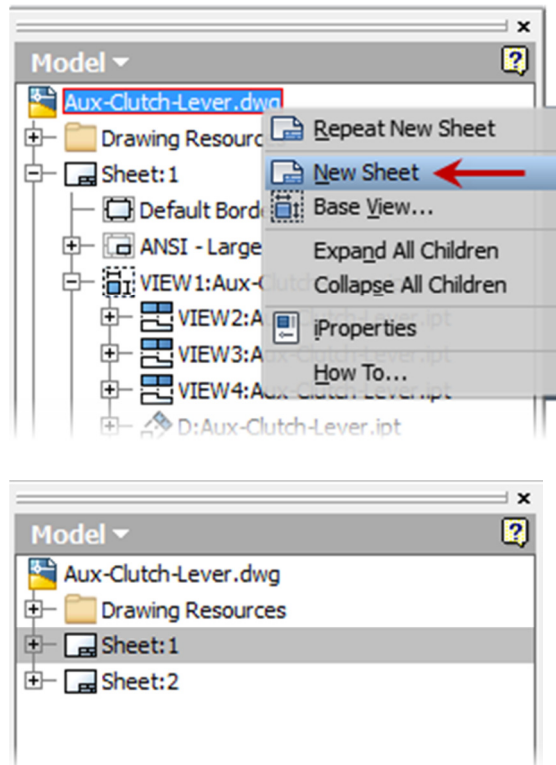
11. Project: Manage Views

In this portion of the project you move, copy, align, and rotate drawing views to prepare a drawing for annotations.

Instructions

45. Create a new sheet:

- Right-click in a blank area of the browser. Click **New Sheet**.
- A new **D** size sheet is created and becomes the active sheet.



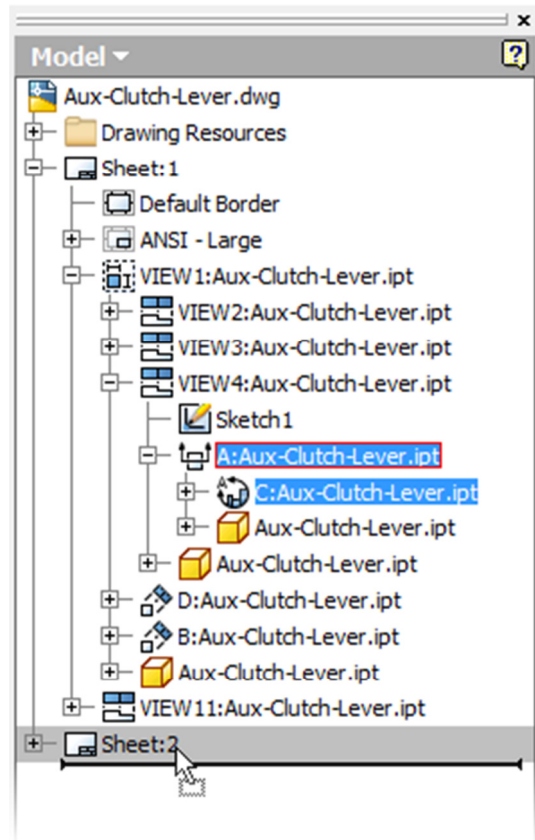
46. In the browser, double-click *Sheet:1* to activate the sheet. Expand *Sheet:1*.

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Project 1 – Drawing View Creation

47. Move two (2) views from *Sheet:1* to *Sheet:2*:

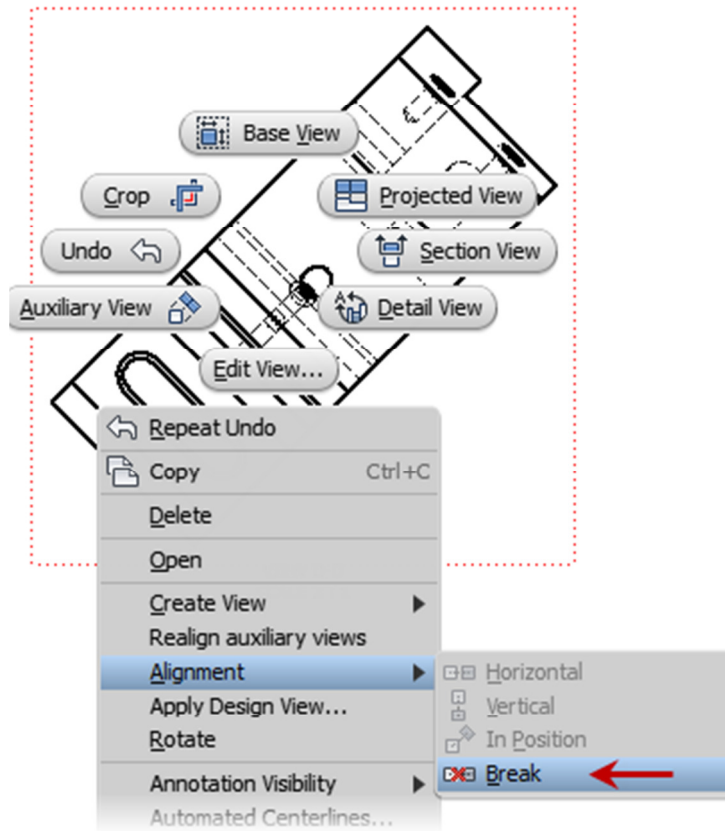
- In the browser, expand *VIEW4* to display the other views associated with it.
- Drag and drop the section view *A:Clutch-Lever-D.ipt* and detail view to *C:Clutch-Lever-D.ipt* *Sheet:2*.



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48. Right-Click on the **Auxiliary View** and select **Break** from the **Alignment** flyout.

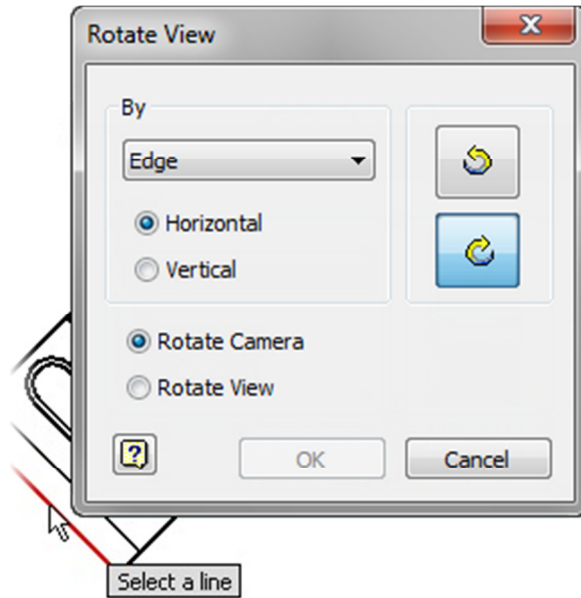


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49. Rotate the auxiliary view:

- In the browser, double-click *Sheet:1*.
- In the browser, expand *View2*.
- Right-click the auxiliary view. Click **Rotate**.
- Select the lower-left diagonal line. In the **Rotate View** dialog box, click **Counter Clockwise**. Click **OK**.

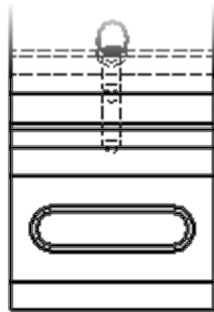


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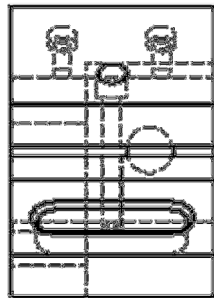
Project 1 – Drawing View Creation

50. Align the auxiliary view to the left view:

- Right-click in the rotated auxiliary view. Click **Alignment | Vertical**.
- Click the left view.



VIEW D-D
SCALE 2 : 1



51. Close all files and save when prompted.