ME 24-688 – Week 6

Project 2 – Dimensions and Annotations

Dimensions and Annotations

After creating drawing views, you can annotate those views with dimensions, hole and thread notes, centerlines, and symbols. Production-ready drawings also typically include revision tables and tags. While traditional annotation methods can be quite tedious, you can quickly and easily include these elements in your drawings using the annotation tools available in Autodesk® Inventor®.

1. About Automatically Placed Dimensions

Overview

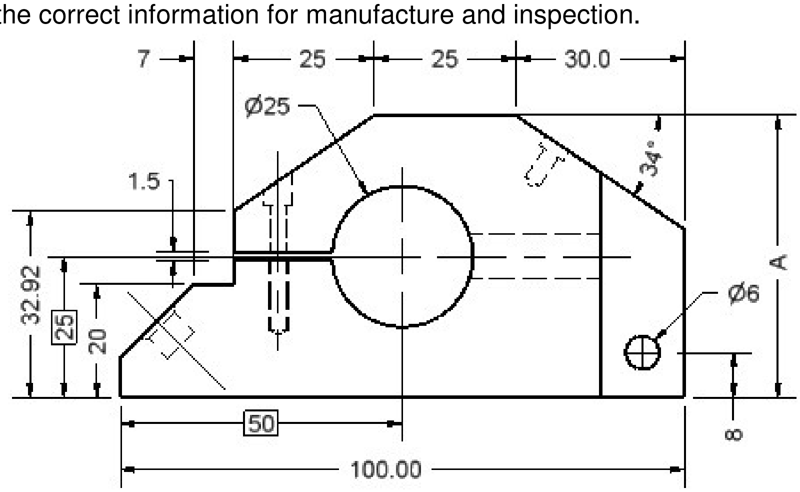
Dimensions are used extensively in the modeling process. Dimensions can be retrieved from the model and reused in drawing views. Retrieved dimensions can be added to or deleted from drawing views at any time, without affecting the model. However, if the default option of enabling part modifications from drawing views is selected at the time the program is installed, editing the value of retrieved dimensions will edit the model.

In the illustration, the model is created and the drawing view shown. During the process of applying dimensions, an engineering change is received. The designer updates the changed dimension, and the part is updated in the drawing view.

Definition of Automatically Placed Dimensions

Automatically placing a dimension in a drawing view is the process of retrieving a dimension that has been applied in the modeling environment and using it to annotate the drawing view in the drawing environment. Because the retrieved dimension is displayed as applied in the modeling environment, it can be edited to meet your drawing standards.

In the illustration, model dimensions are retrieved from the modeling environment and used to dimension a drawing view in the drawing environment. The appearance of the dimensions is then edited to convey the correct information for manufacture and inspection.

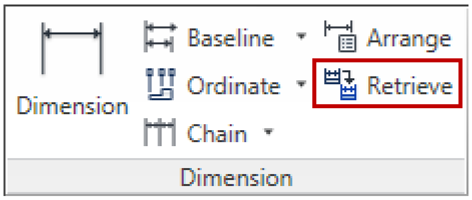


ME 24-688 – Week 6

Project 2 – Dimensions and Annotations

Access

Ribbon: **Annotate tab | Dimension panel | Retrieve**

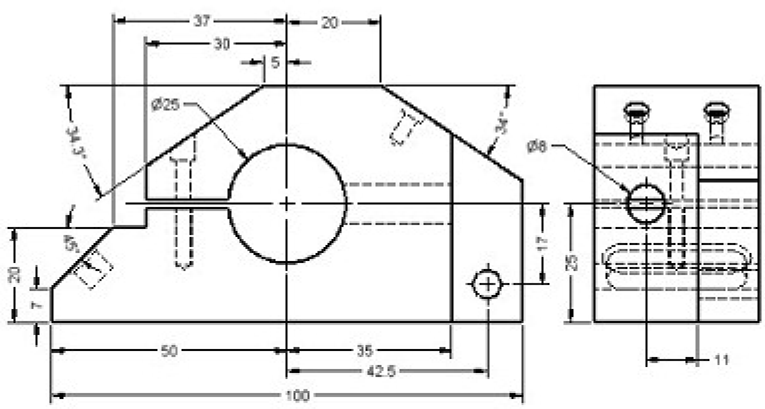


Overflow Menu: **Retrieve Dimensions**

2. Creating General Dimensions

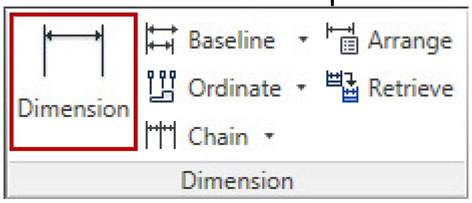
The **General Dimension** tool can place many different types of dimensions, depending on the geometry selected. Selecting two parallel lines results in horizontal, vertical, or aligned dimensions; selecting two nonparallel lines results in angle dimensions; and selecting an arc or circle results in radial or diameter dimensions. To obtain a horizontal or vertical dimension between two nonparallel lines, you select one line and the endpoint of the other line, or select two endpoints.

In the illustration, general dimensions are placed on the drawing to define the part.



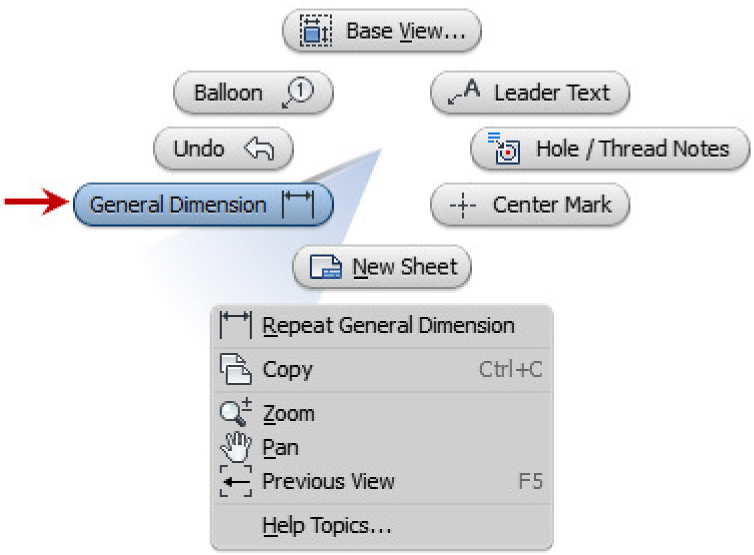
Access

Ribbon: **Annotate tab | Dimension panel | Dimension**



Keyboard: **D**

Marking Menu: **General Dimension**



Horizontal: Creates a horizontal dimension based on the points or segments selected.

Vertical: Creates a vertical dimension based on the points or segments selected.

Aligned: Creates a linear dimension perpendicular to the points or segments selected.

3. Working with Hole and Thread Notes

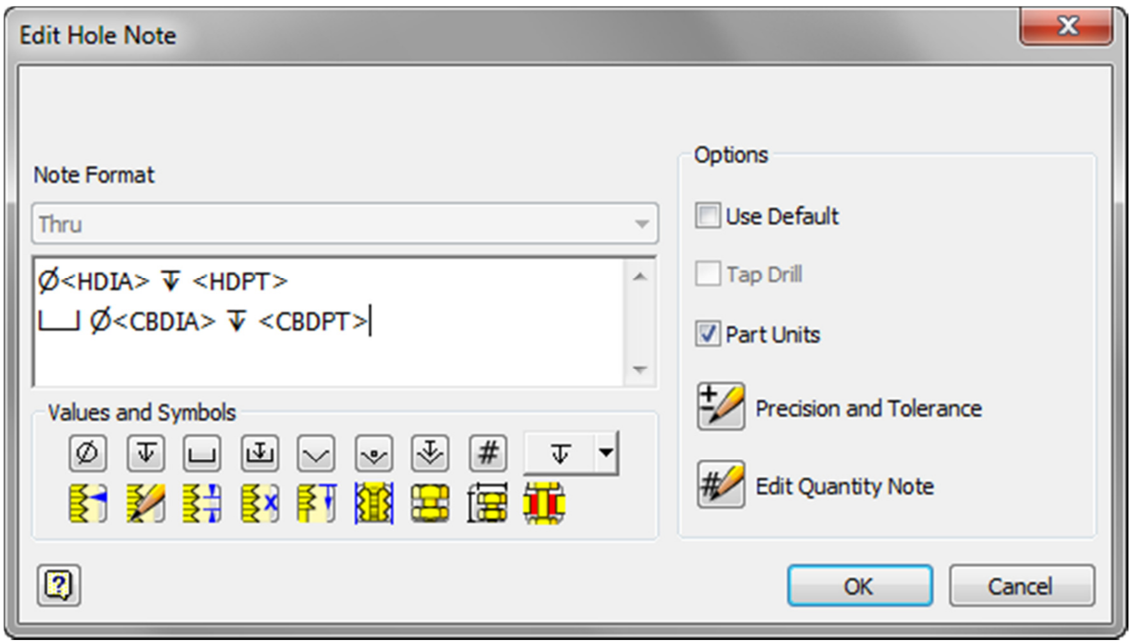
Holes in production drawings are defined by the notes attached to them. A part may contain several holes of the same size; however, because of their functions, different processes may be required to create them. Hole and thread notes define the process to create the holes, and therefore the quality and function of the part. The procedure for adding and editing a hole note is very similar to the procedure for adding and editing diameter dimensions.

ME 24-688 – Week 6

Project 2 – Dimensions and Annotations

Edit Hole Note Dialog Box

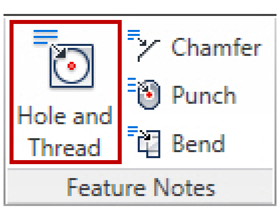
You use the options in the **Edit Hole Note** dialog box to edit the selected hole note. You can use the text box to add static text in the note, to add specific symbols, or to add dynamic value codes. You can also change the display options to include the tap drill information; the precision, tolerance, and unit type; and the method for displaying the quantity of holes.



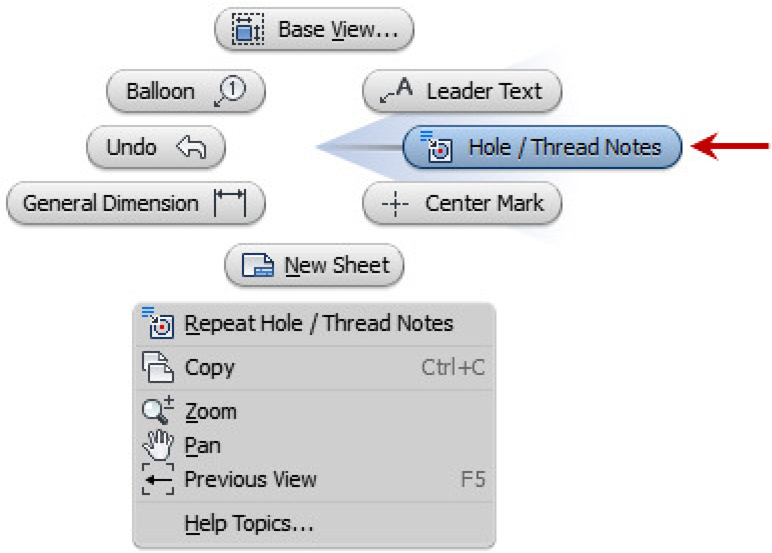
When you use the **Edit Hole Note** dialog box to edit a hole note, the changes apply only to the selected note. They do not affect the other hole notes in the drawing, even if they are on the same types of holes.

Access

Ribbon: **Annotate tab | Features panel | Hole and Thread**



Marking Menu: **Hole** / **Thread Notes**



Holes You Can Annotate with Notes

You can select the types of holes for annotation:

- Features created with the Hole tool
- Extrude-cut circles and circular sheet metal cuts
- Voids in extrude-join operations
- Sheet metal features
- Holes in features

Guidelines

- You control the symbols and text contained in a hole note with the **Dimension Style** settings**.**
- You can include a variety of note data on a hole based on the combination of hole type and thread type.
- In order for the **Hole/Thread Notes** tool to add annotations to a drawing view, the hole must have been created using one of the specified methods.
- You can change the display of an existing hole note by changing its dimension style.
- You can modify the annotation information within the hole note using the **Edit Hole Note** dialog box.
- You can add text characteristics and additional text and symbols to a hole note using the **Format Text** dialog box.
- Editing a hole note in a drawing does not change the initial standard on which that note was based.

ME 24-688 – Week 6

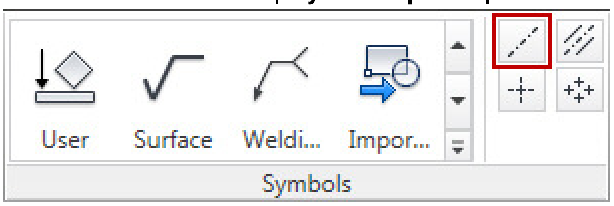
Project 2 – Dimensions and Annotations

4. Creating Centerlines, Symbols, and Leaders Introduction

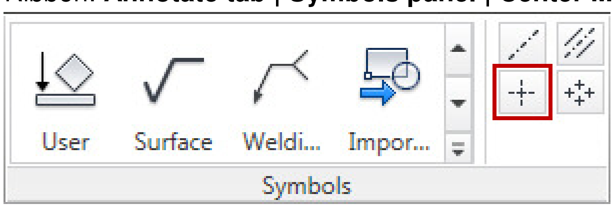
The use of centerlines and center marks is critical in the interpretation of symmetrical and cylindrical features in drawings. Symbols aid in defining the manufacturing processes used in creating parts and assemblies. Leaders provide information about the text, symbol, or dimension attached to specific areas on a feature or object.

Access

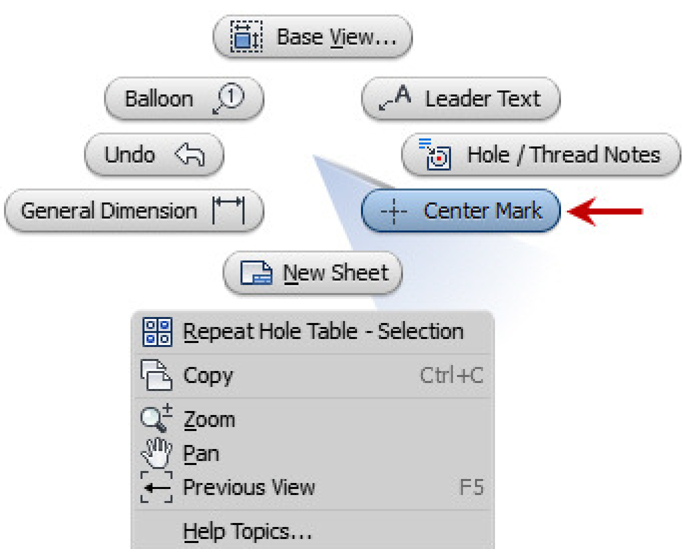
Centerline:
Ribbon: **Annotate tab | Symbols panel | Centerline**



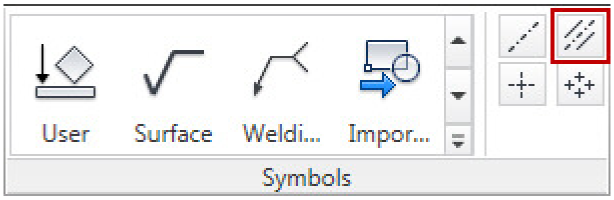
Center Mark:
Ribbon: **Annotate tab | Symbols panel | Center Mark**



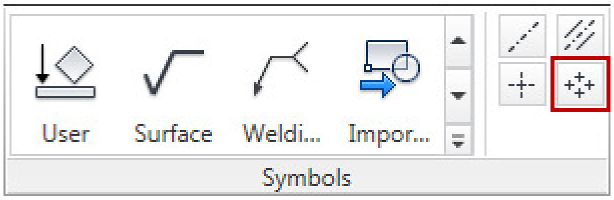
Marking Menu: **Center Mark**



Centerline Bisector:
Ribbon: **Annotate tab | Symbols panel | Centerline Bisector**



Centered Pattern:
Ribbon: **Annotate tab | Symbols panel | Centered Pattern**



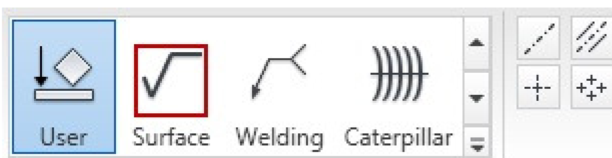
5. Documenting Views with Symbols

Symbols are key elements in the creation of production-ready drawings. Once you have determined the symbol needed to define the drawing, you select the tool and apply the symbol. There are many different symbol tools, but they typically follow a similar procedure.

Access

Surface Texture:

Ribbon: **Annotate tab | Symbols panel**



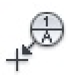
Feature Control Frame:

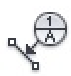
Ribbon: **Annotate tab | Symbols panel**

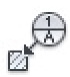
Datum Identifier:

Ribbon: **Annotate tab | Symbols panel**

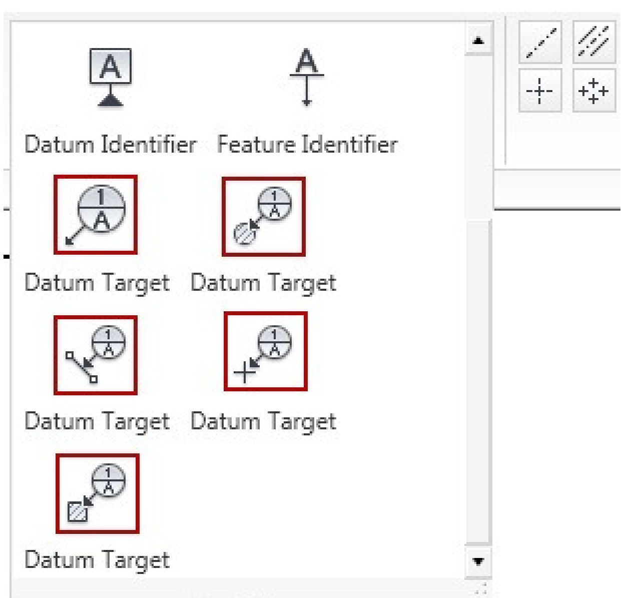
Datum Target:







Ribbon: **Annotate tab | Symbols panel**

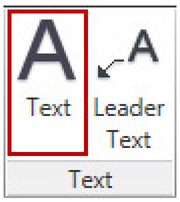


6. Adding Leaders and Text

You use a **leader with text** to define a certain condition or specification that cannot be defined with a standard symbol. **Leaders** are **associative** to the feature that they are attached to. If that feature changes location, the leader will change location also. If the feature is deleted, the leader will be deleted from the view.

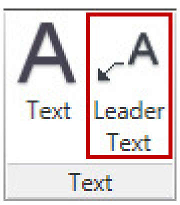
Access

Text :
Ribbon: **Annotate tab | Text panel | Text**



Keyboard: **T**

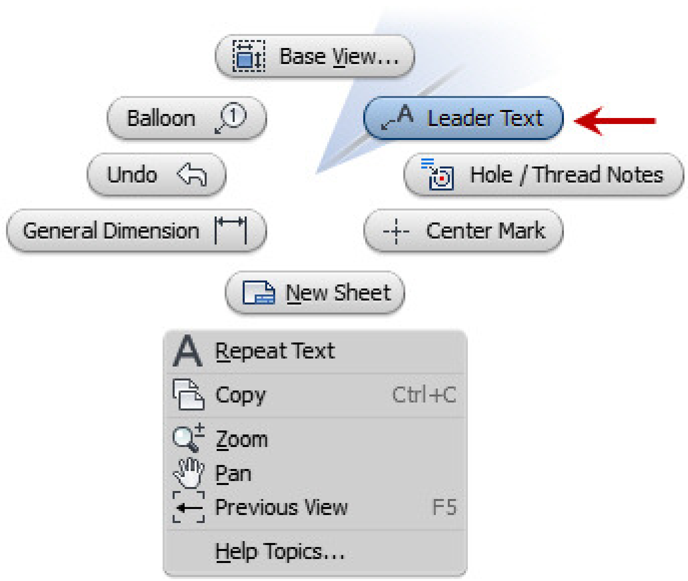
Leader Text:
Ribbon: **Annotate tab | Text panel | Leader Text**



Toolbar: **Drawing Annotation Panel**

Keyboard: **Ctrl+Shift+T**

Marking Menu: **Leader Text**

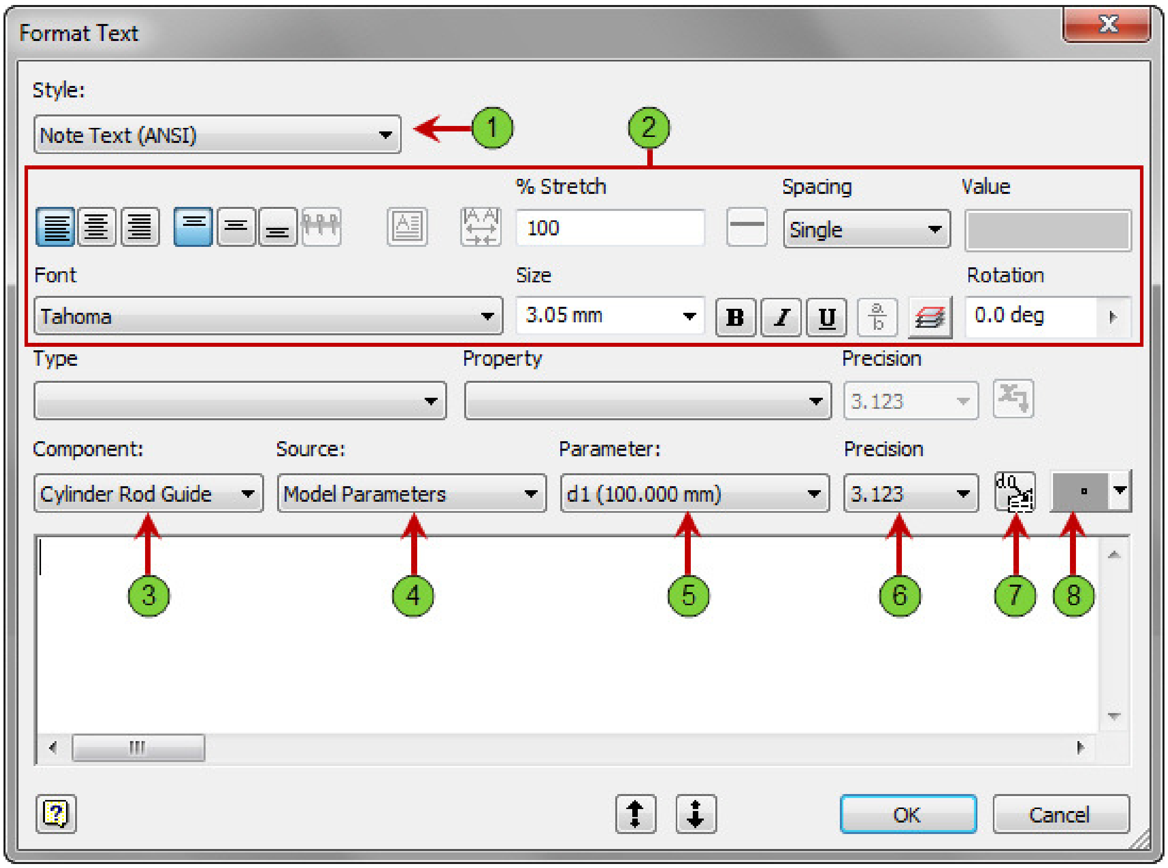


Base View...
Balloon
Leader Text
Undo
Hole / Thread Notes
General Dimension
Center Mark
New Sheet
Repeat Text
Copy
Ctrl+C
Zoom
Pan
Previous View
F5
Help Topics...

Notes and Leaders

You use the **Text** and **Leader Text tools** to add **notes and leaders** to drawing views. When you use the **Text tool** to place **paragraph-style text** on the sheet, the **Leader Text tool** attaches a **leader with text** to the geometry within the view.

The illustration shows the **Format Text dialog box**, which you use to add text to your drawings.



1 **Style**: Select a **text style** for the text or accept the **default text style** listed.

2 **Text Formatting:** Adjust the **text formatting options** such as justification, text size, color, and width as required.

3 **Component** (**Optional**): Select the component to be used for parameters.

4 **Source** (**Optional**): Select **Model** Parameters or **User** Parameters.

5 **Parameter**(**Optional**): Select the parameter to use in the text.

6 **Precision** (**Optional**): Enter a **precision** for the parameter value.

7 **d0 Button**(**Optional**): Click to add the selected parameter to the text window.

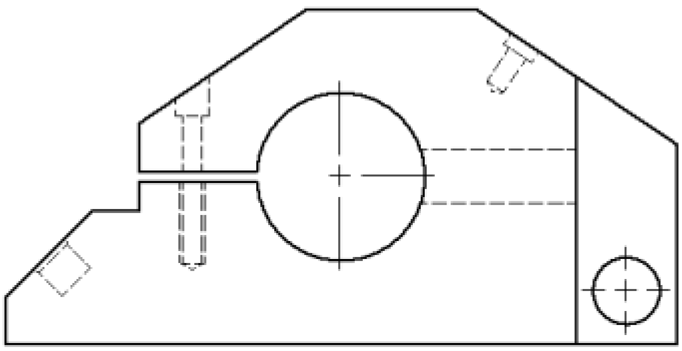
8 **Symbols Flyout:** Select a special **symbol** to insert into the text.

7. Project: Dimension a Drawing View

In this portion of the project, you dimension a view by retrieving the dimensions that were used to create the model. You then edit one of the dimensions to update the drawing and the model.

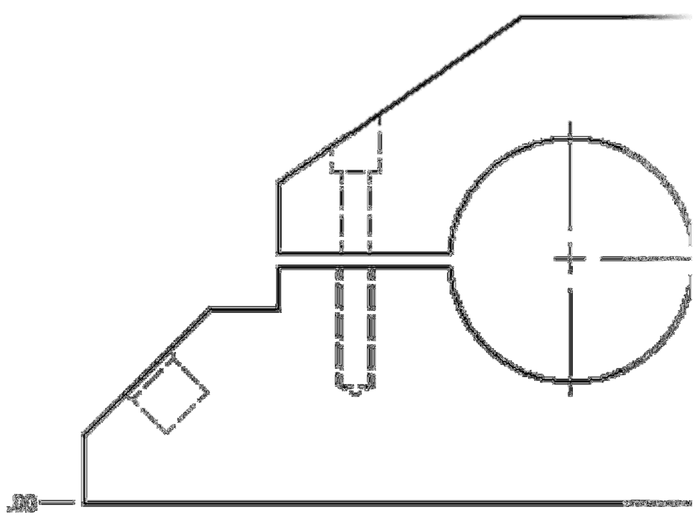
Instructions

1. Open *Dimensioning Aux-Clutch-Lever.dwg.*
2. Add center marks to the front view.



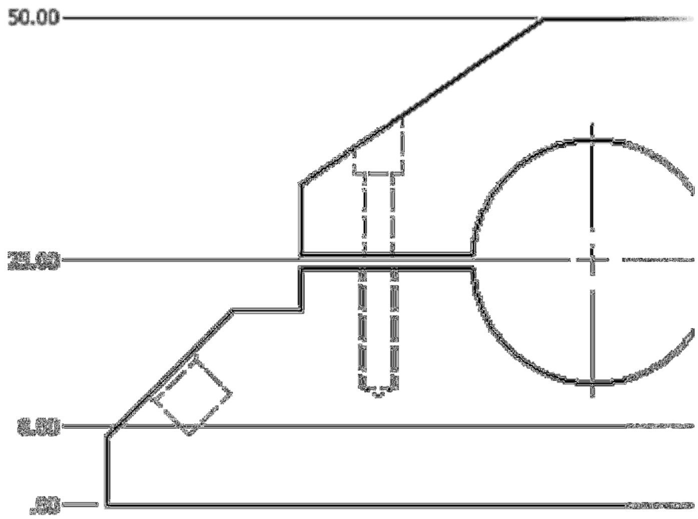
- Click **Annotate tab | Symbols panel | Centerline Mark**.
- In the Front view, select the cylindrical edge of both holes in the front view.
- Right-click and click **Done**.

3. Define the zero point for the ordinate dimension set:



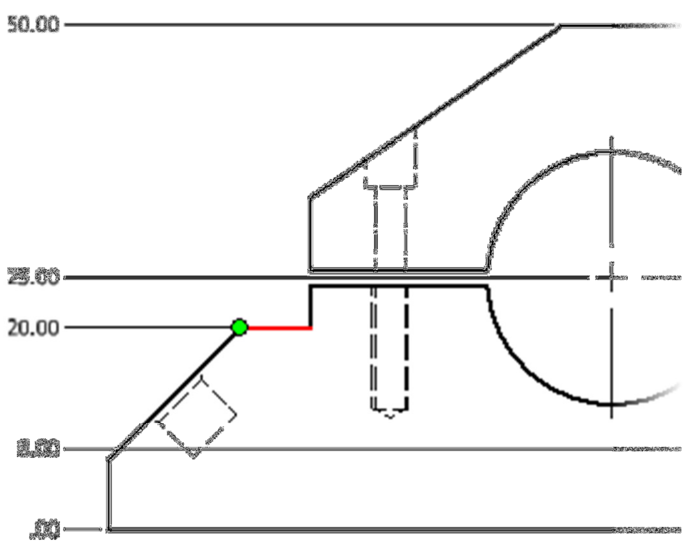
- Click **Annotate tab |Dimension panel | Ordinate Set.**
- Click in the left corner of the part.
- Right-click. Click **Continue**.
- Click to the left of the view to place the dimension as shown in the illustration.

4. Place ordinate dimensions:



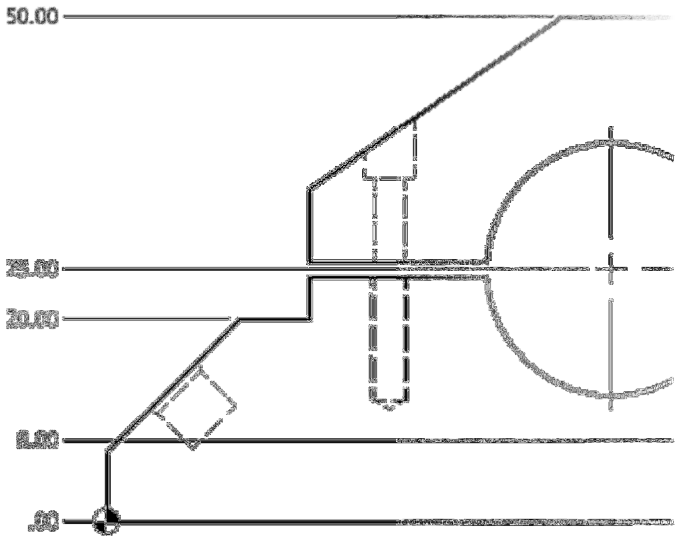
- Select the two (2) Center marks, the top edge.
- Right-click and click **Create**.

5. Add an ordinate dimension to the set:



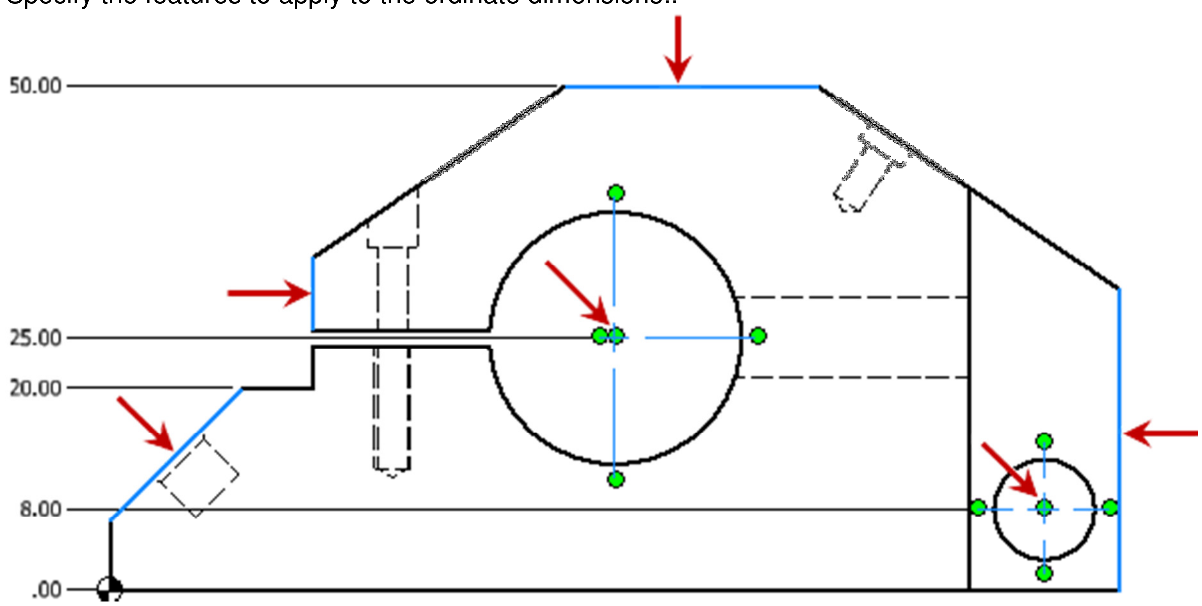
- Move the cursor to any dimension in the ordinate set.
- Right-click and click **Add Member**.
- Select the edge as shown.

6. Define the zero origin for the ordinate dimensions in the Y direction:



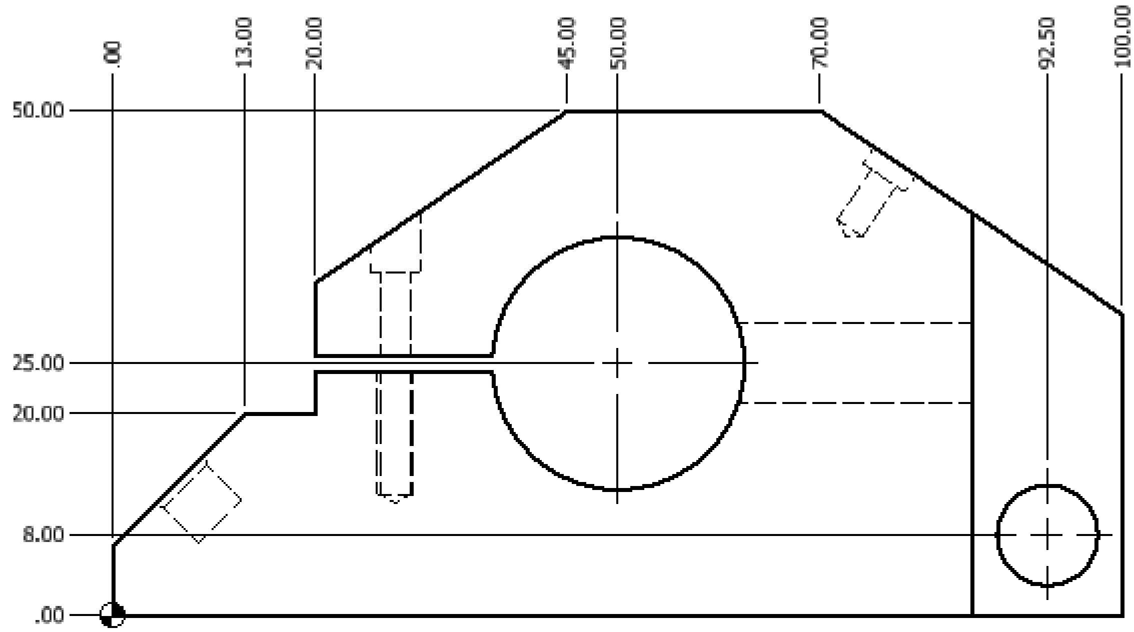
- Click **Annotate tab | Dimension panel | Ordinate**.
- Select the Front view.
- Click in the left corner of the part.
- Right-click in the graphics window and click **Continue**.
-

7. Specify the features to apply to the ordinate dimensions::



- Place the ordinate dimensions to the right of the view.
- Right-click. Click **Done**.

8. Place the ordinate dimensions:



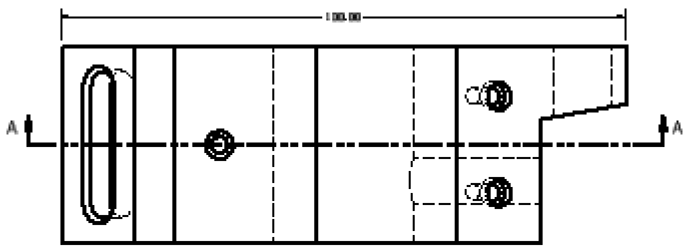
- Right-click in the graphics window and click **Continue**.
- Place the ordinate dimensions to the right of the view.
- Right-click. Click **Done**.

8. Project: Create General Dimensions

In this portion of the project, you add horizontal, vertical, diameter, and angle dimensions to a drawing. You also create a radial dimension and convert it to a diametric dimension.

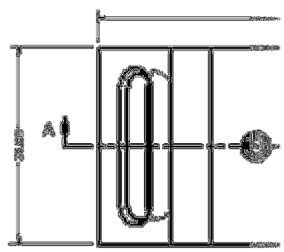
Instructions

9. Create a horizontal dimension:



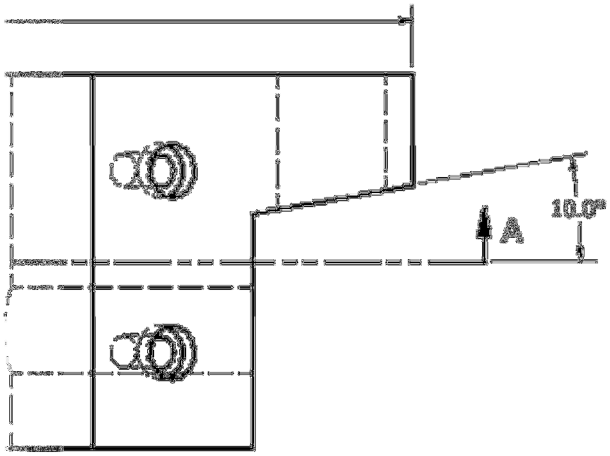
- Click **Annotate tab | Dimension panel | Dimension**.
- In the Top View, click the two vertical lines at each end of the part.
- Place the horizontal dimension above the part.

10. Create a vertical dimension:



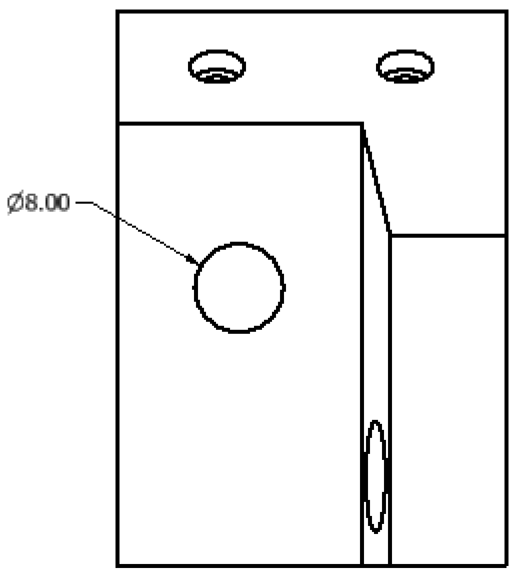
- With the **General Dimension** tool still active, select the bottom line in the top view.
- Select the top line in the left view.
- Place the vertical dimension next to the view.

11. Create an angle dimension:



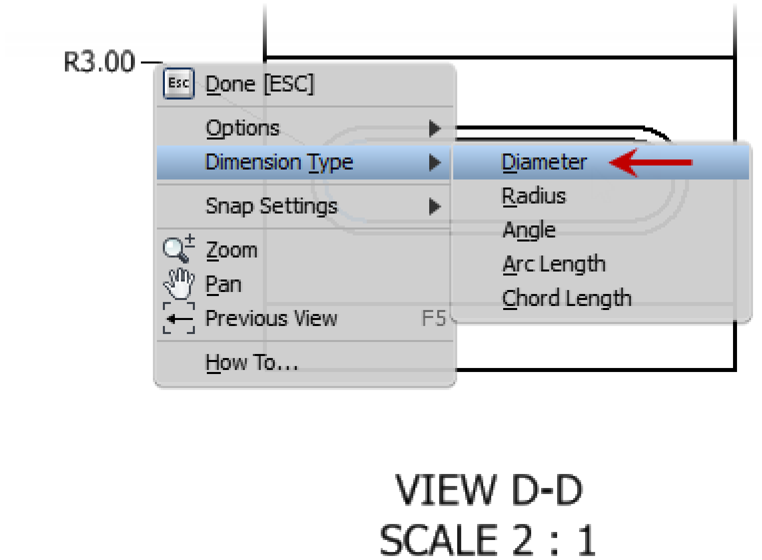
- With the **General Dimension** tool still active, select the section line in the top view, then select the line above it.
- Place the angle dimension.

12. Create a diameter dimension:



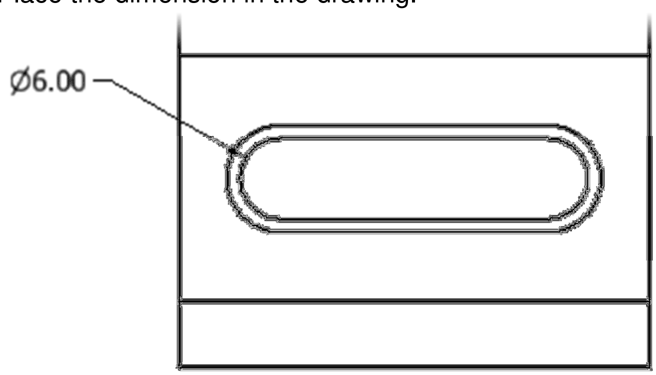
- With the **General Dimension** tool still active, select the **circle** in the right view.
- Click to place the dimension.

13. Create a radial dimension and change to a diameter dimension:

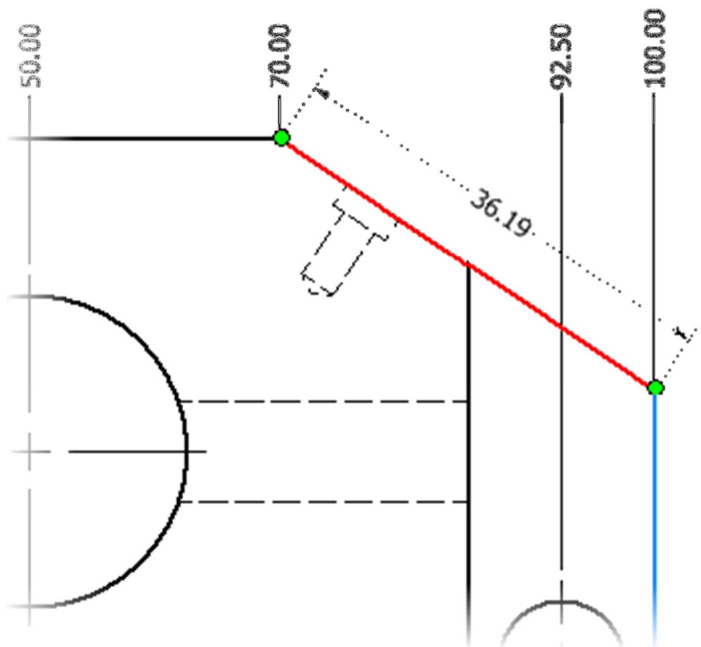


- With the **General Dimension** tool still active, click the left radius on the slot in auxillary **View D-D**.
- Right-click and click **Dimension Type | Diameter**.

14. Place the dimension in the drawing.

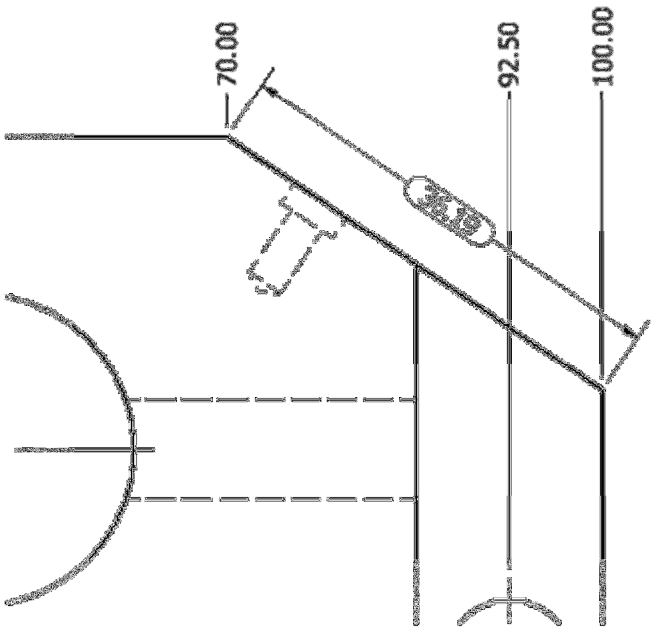


15. Create an aligned dimension:

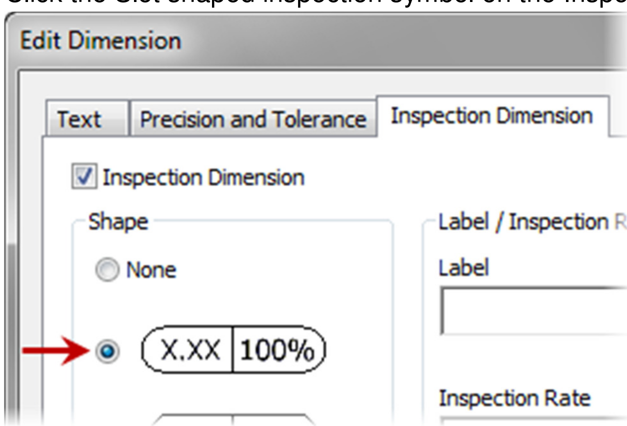


- With the **General Dimension** tool still active, select the angled line on the left side of the front view.
- Position the cursor as shown to automatically orient the dimension to be an aligned dimension.

16. Set as an **Inspection Dimension** and click to place the aligned dimension.



- Click the Slot shaped inspection symbol on the Inspection Dimension tab



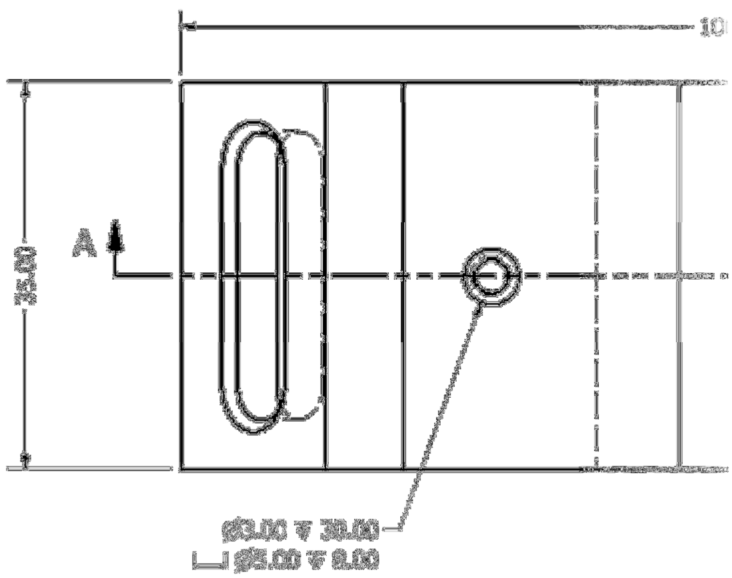
- Click to place the dimension.
- Right-Click and select **Done**

9. Project: Create and Edit Hole Notes

In this portion of the project, you create and edit hole notes to supply the machine shop with data for CNC programming

Instructions

17. To add a hole note:



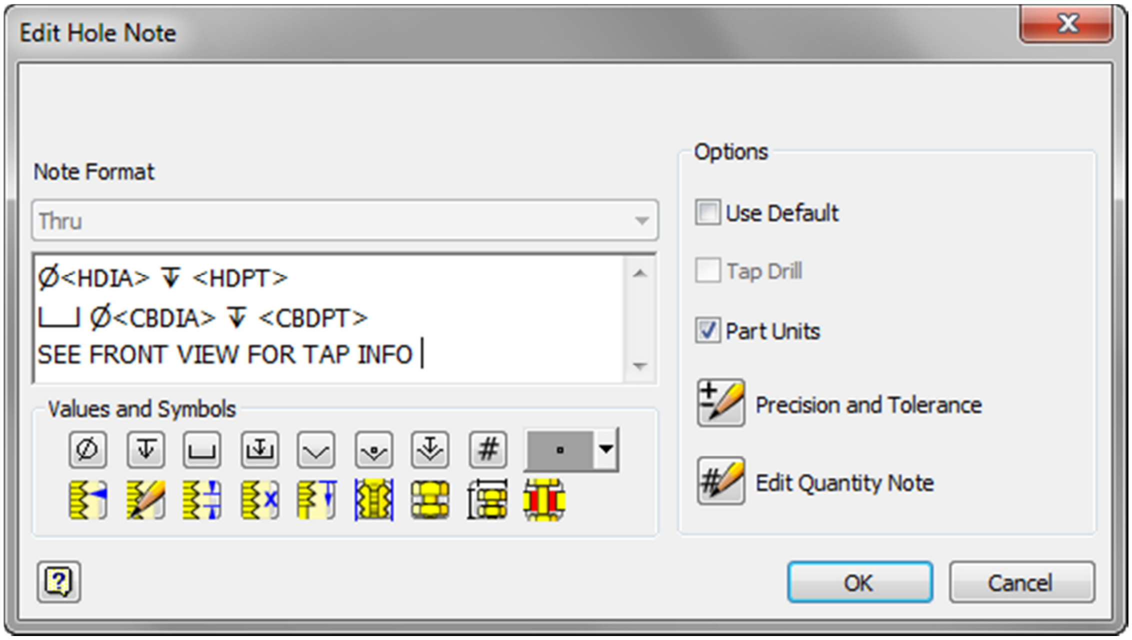
- Zoom in on the Top view.
- Click **Annotate tab | Feature Notes | Hole and Thread**
- Select and place the hole note for counterbored hole as shown.
- Right-click, click **Done**

18. To access the hole note editor:

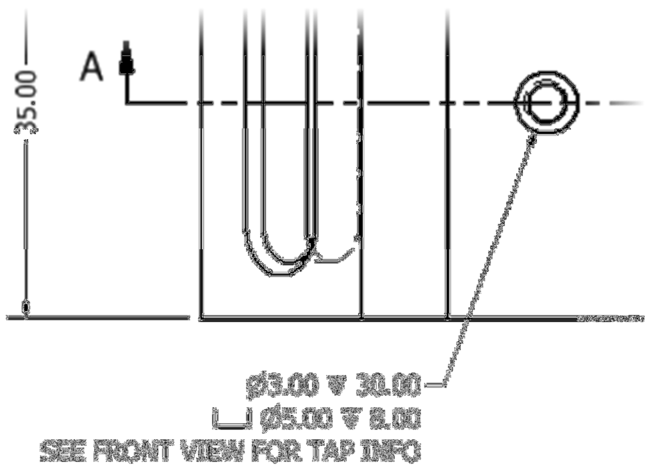
- Right-click the Ø3.00 ↧ 30.00
  ⌴ Ø5.00 ↧ 8.00 note.
- Click **Edit Hole Note**

19. To add the tap information at the end of the note, take the actions:

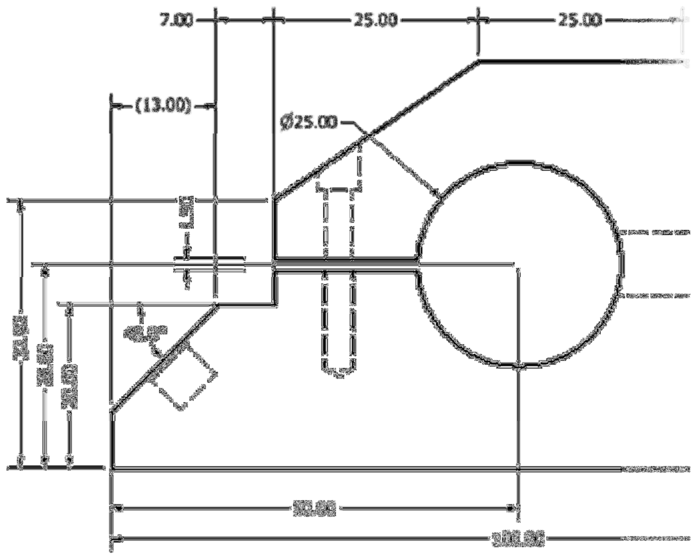
- Add a line a Type: **SEE FRONT VIEW FOR TAP INFO**



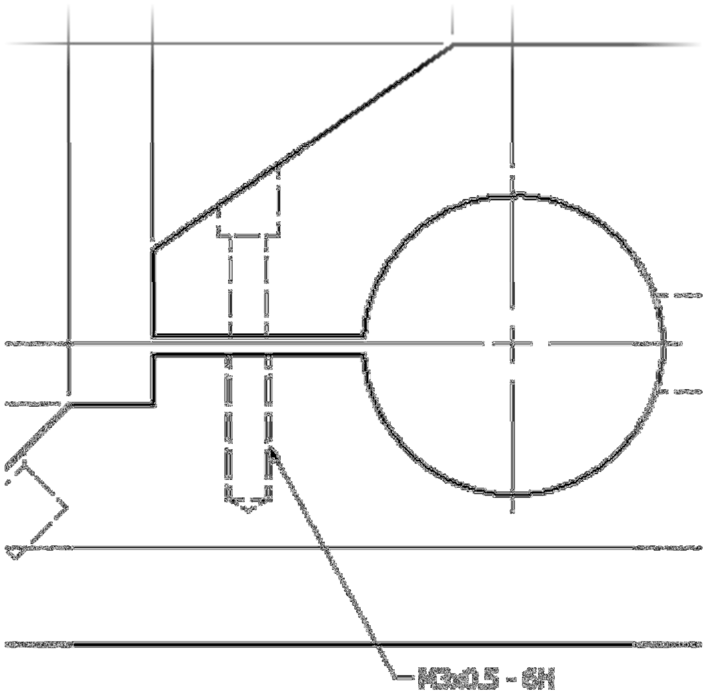
20. Click **OK**. The note is modified as shown.



21. Zoom in on the Front view.



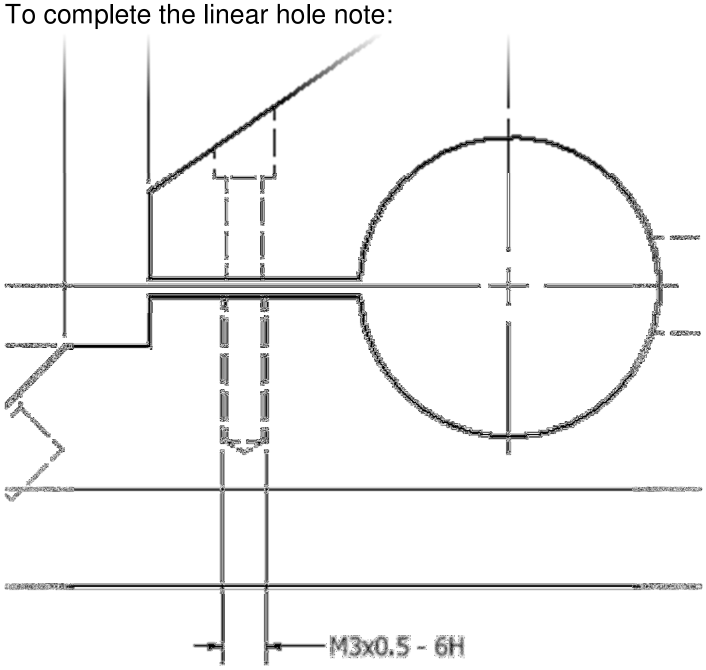
22. To begin to add a linear hole note:



- Select one edge of the threaded hole, as shown.

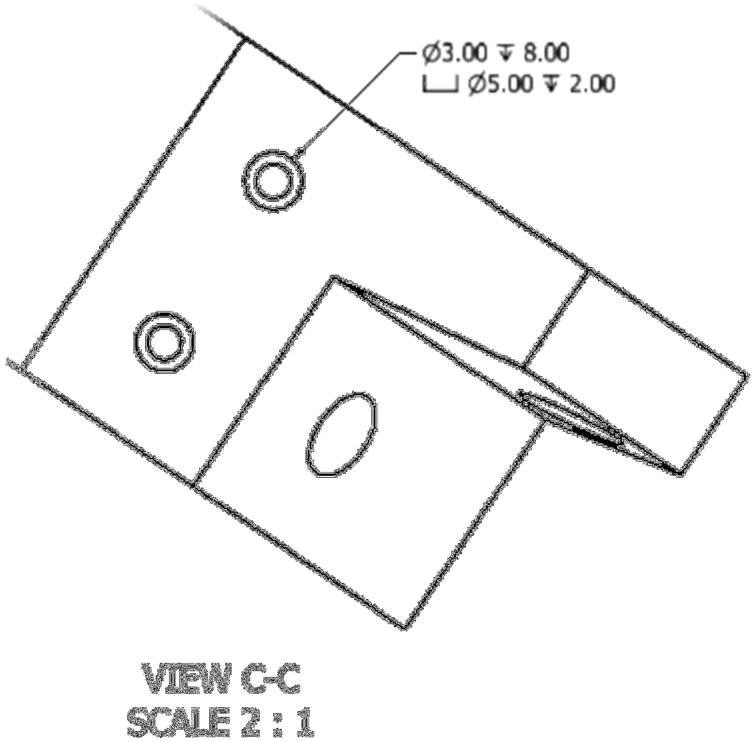
Notice the leader and the hole note text.

23. To complete the linear hole note:



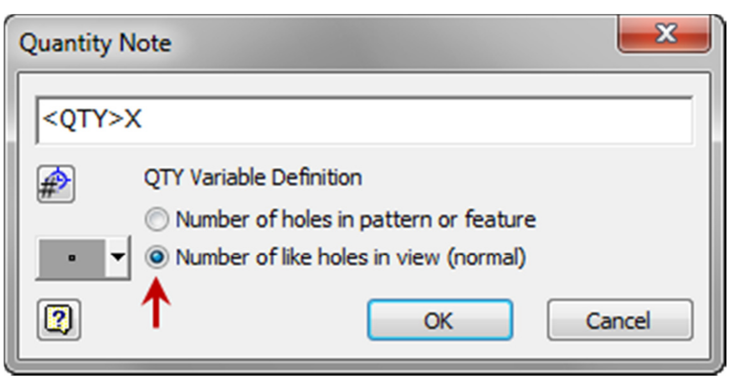
- Click the lower edge of the threaded hole.
- Click to place the hole note.

24. Add the note as shown in the illustration. Exit the tool after placing the note.



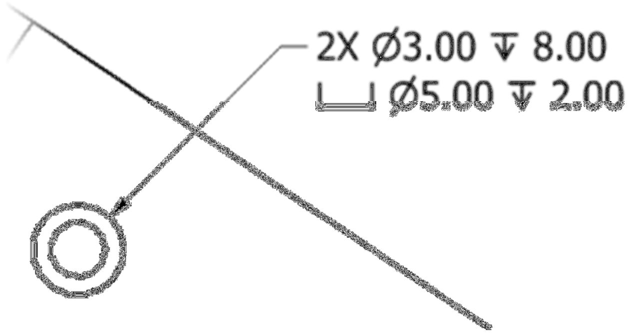
25. Right-click the Ø3.00 ↧ 8.00
⌴ Ø5.00 ↧ 2.00 note. Click **Edit Hole Note.**

26. In the Edit Hole Note dialog box:



- Under **Options**, click **Edit Quantity Note**.
- In the **Quantity Note** dialog box, select **Number of Like Holes in View (Normal).**
- Click **OK**.

27. Next, edit the hole note to have the quantity of like holes displayed in the note.



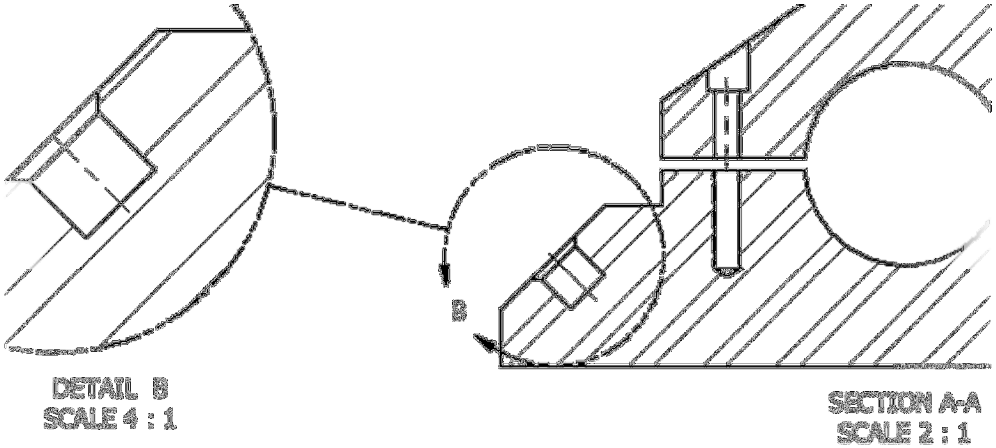
- Position the cursor to the left of the text in the **Edit Hole Note** dialog box.
- Click the **#** icon in the **Values and Symbols category**.
- Click **OK**.

10. Project: Add Centerlines, Center Marks, and Symbols

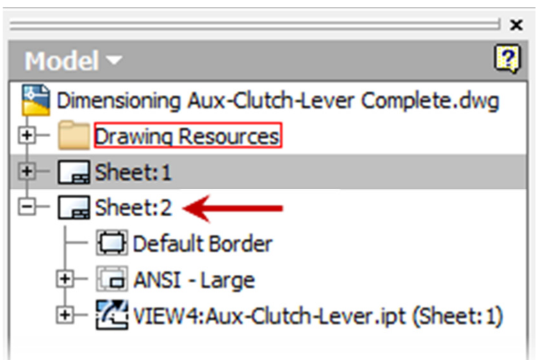
In this portion of the project, you annotate a drawing of a cylinder rod guide with centerlines, center marks, and symbols.

Instructions

28. Add centerlines to the section and detail view.

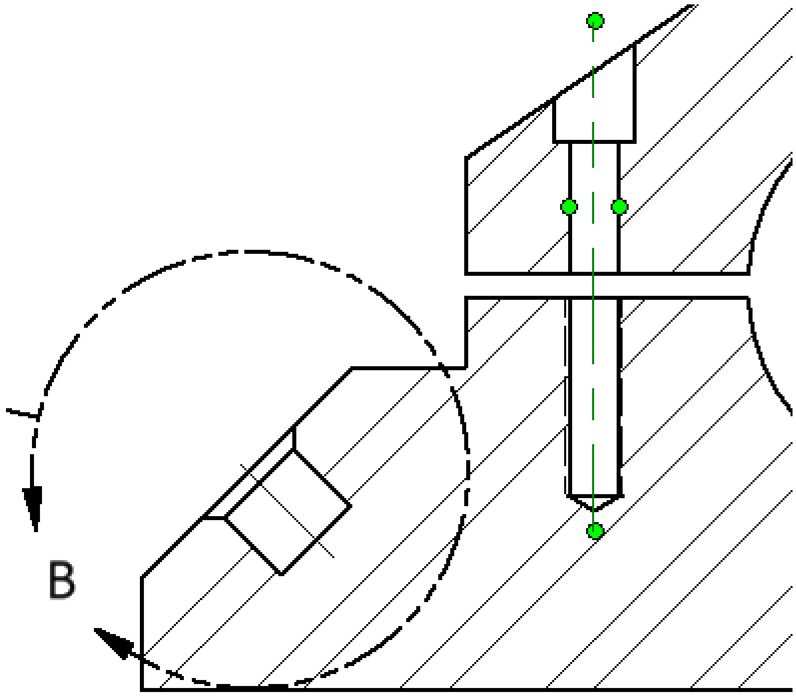


- Double Click on Sheet:2 to activate the sheet



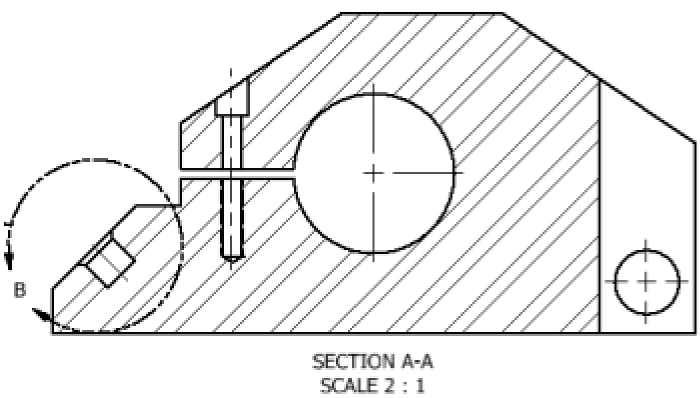
- Click **Annotate tab | Symbols panel | Centerline Bisector**.
- In the section view, select the outer symmetrical lines for the cylinder.
- Select the symmetrical lines defining the bolt and guide pin holes.
- Right-click and click **Done**.
- Add centerlines to the section and detail view.

29. Adjust the centerline length.



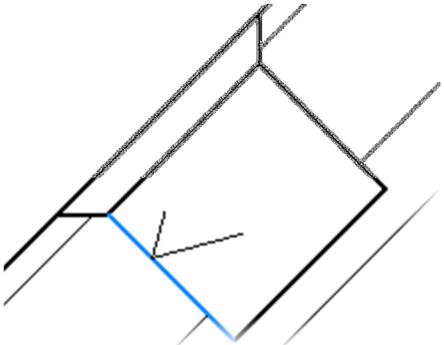
- Move the cursor to the end of the centerline.
- Select the grip and drag past the end of the part on one end and past the end of the hole on the other.

30. Add center marks to the clearance hole and clamping hole:



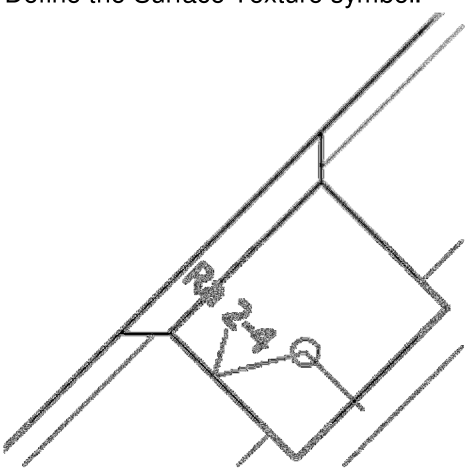
- Click **Annotate tab | Symbols panel | Center Mark.**
- Select the clearance hole and the clamping hole.
- Right-click. Click **Done.**

31. Locate and define a surface texture symbol on the inner surface of the slot in the detailed view:



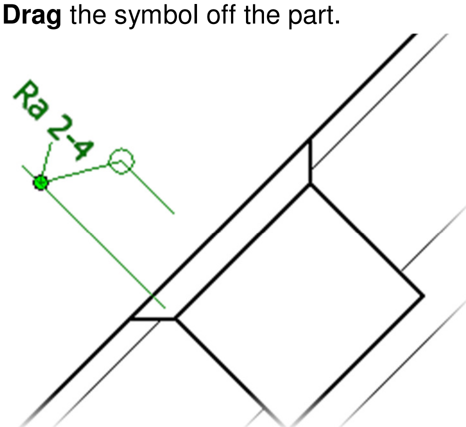
- Click **Annotate tab | Symbols panel | Surface Texture.**
- In the section view, click the inner surface of the slot.
- Right-click and click **Continue**

32. Define the Surface Texture symbol.



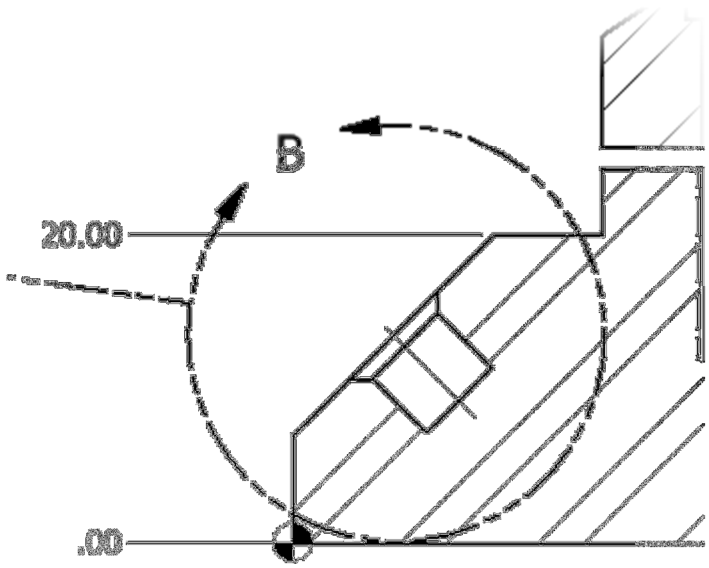
- In the **Surface Texture** dialog box, under **Miscellaneous**, click **All-around**.
- In the **A' field**, enter **Ra 2-4**
- Click **OK**.
- Right-click in the graphics window, click **Done.**

33. **Drag** the symbol off the part.

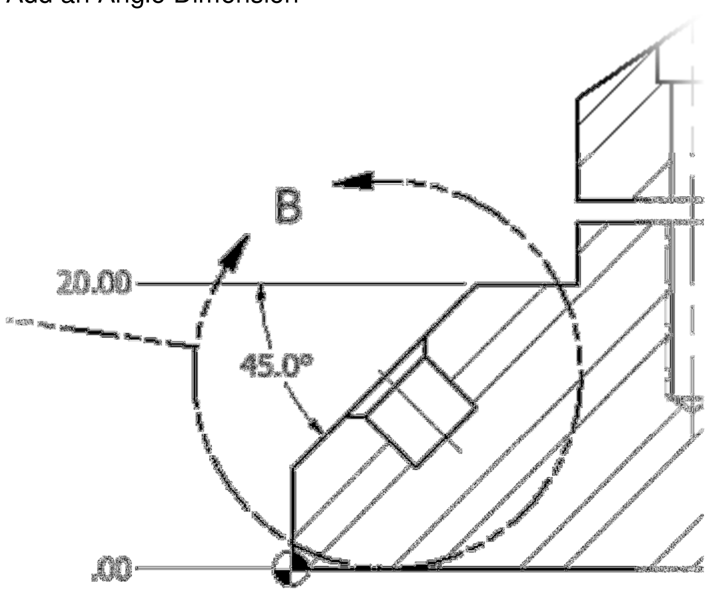


Project 2 – Dimensions and Annotations

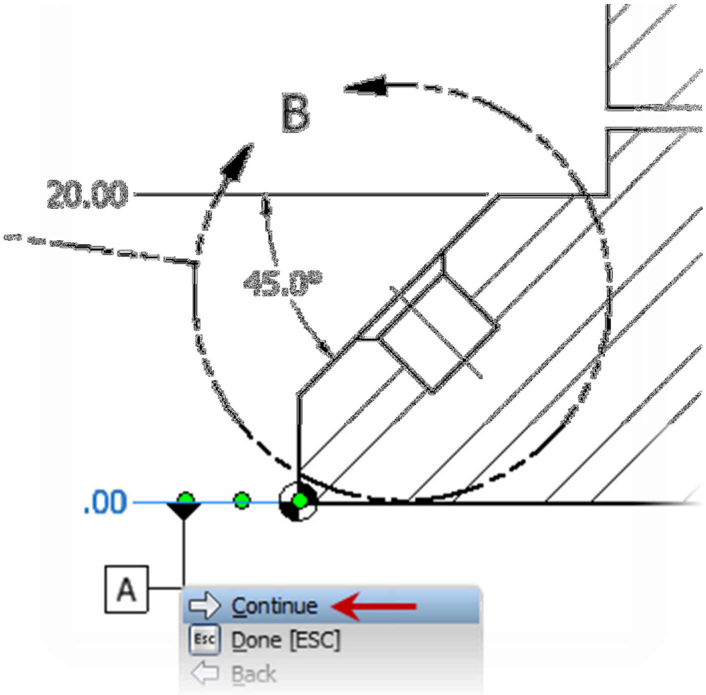
34. Add Ordinate Dimensions.



35. Add an Angle Dimension

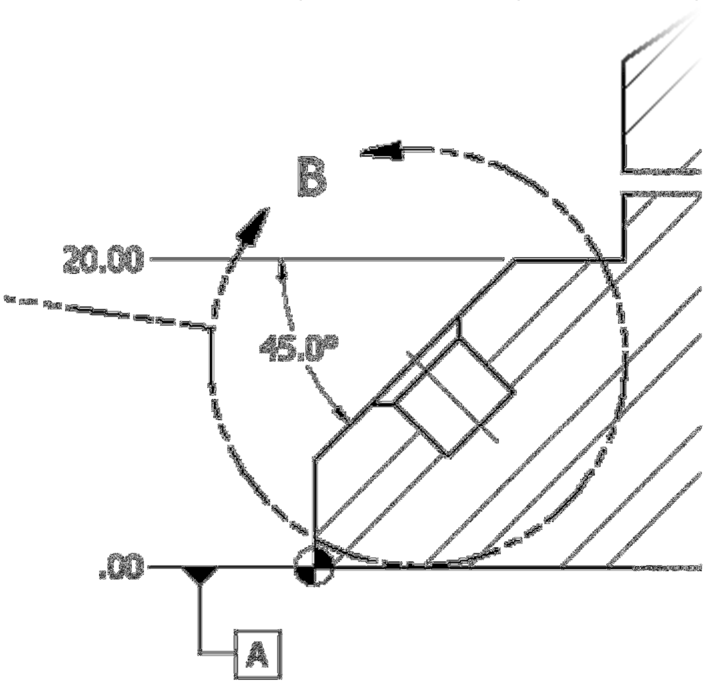


36. Add a Datum Identifier symbol.

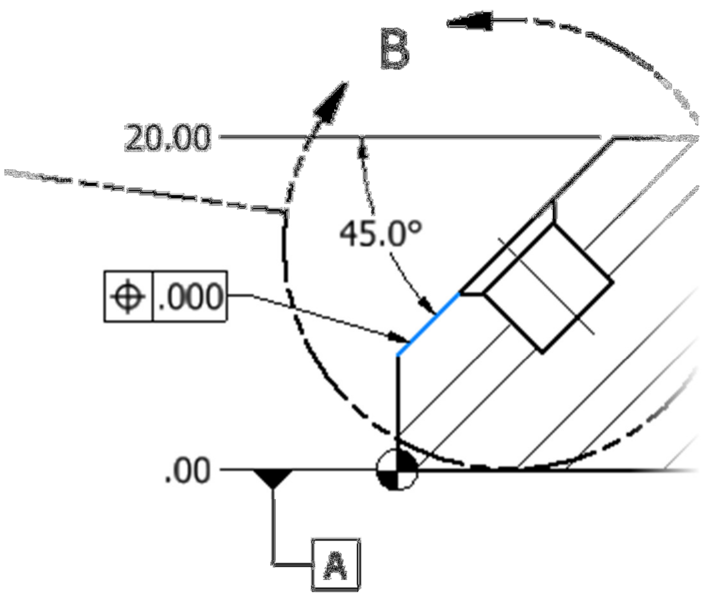


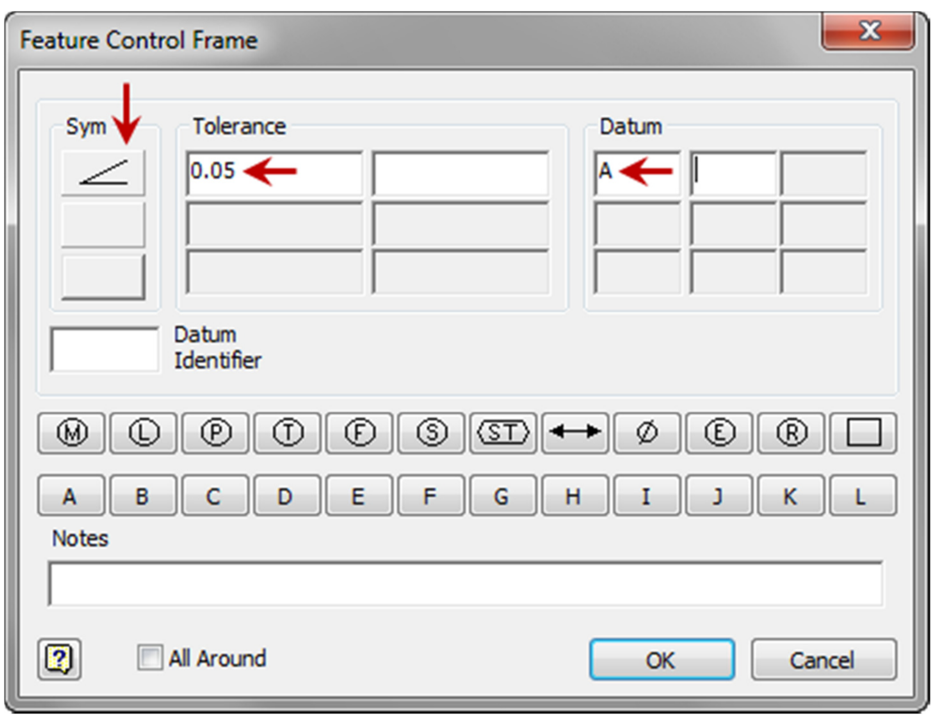
- Click **Annotate tab | Symbols panel | Datum Identifier**. (Expand the available options).
- Click the arrow point on the *.00* dimension added in the previous step.
- Click a point below the first point.
- Right-click and click **Continue**.

37. In the **Format Text** dialog box, click OK. Right-click in the graphics window and click **Done**.



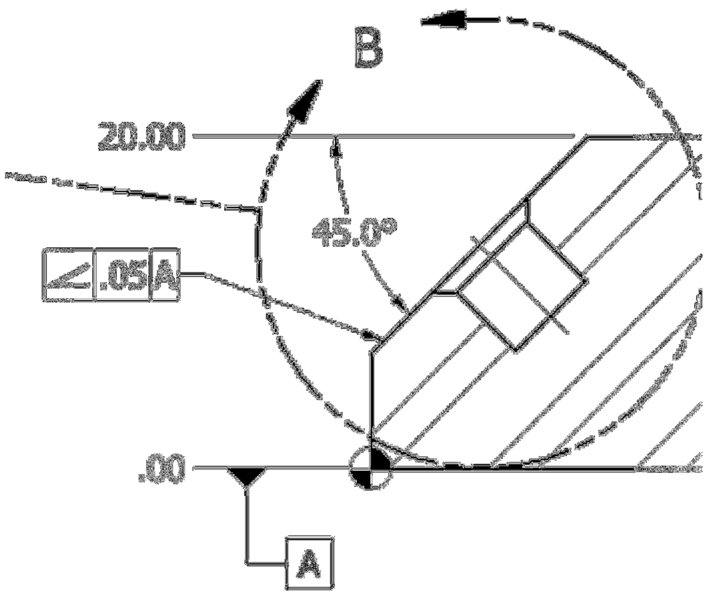
38. Add a feature control frame.



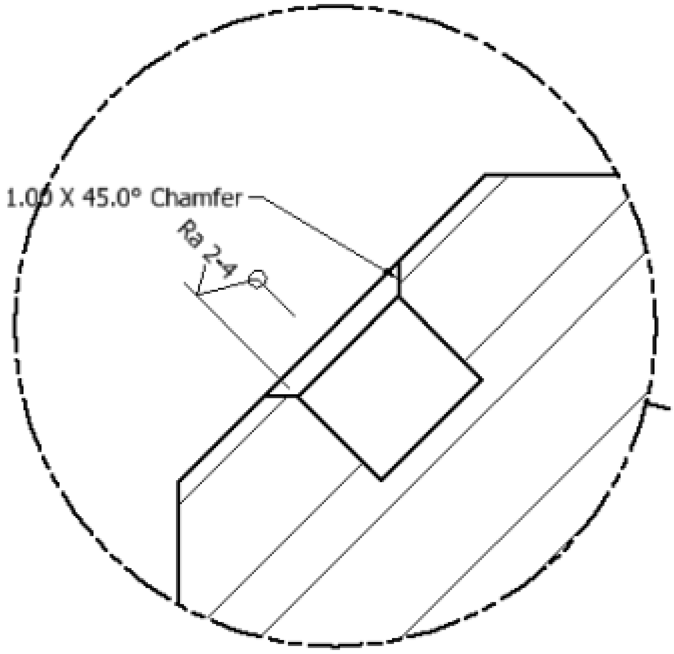


- Click **Annotate tab | Symbols panel | Feature Control Frame.**
- Click the angled edge. Click to position the **Feature Control Frame.** Right-click and click **Continue**.
- In the **Feature Control Frame** dialog box, enter the data as illustrated.

39. In the **Feature Control Frame** dialog box, click **OK**. Right-click in the graphics window and click **Done**.

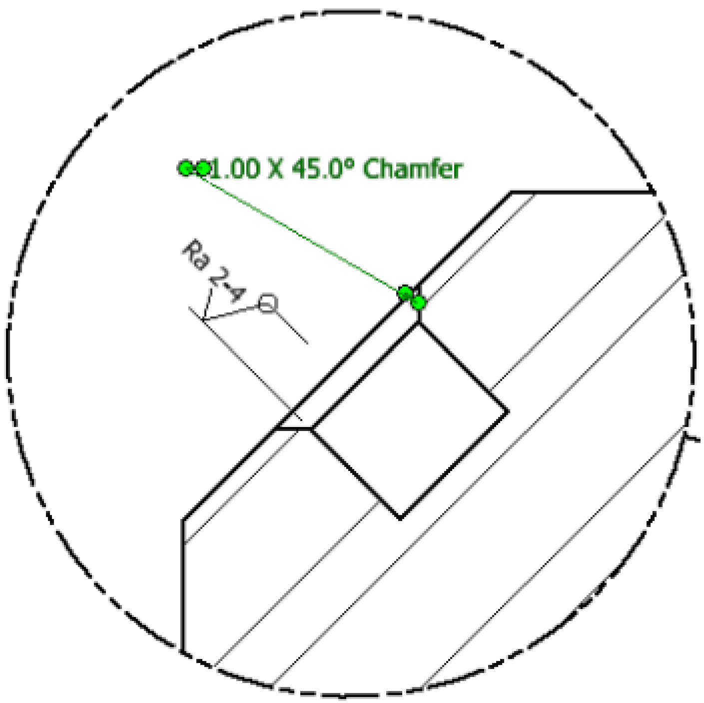


40. Add a Chamfer note:



- Click **Annotate tab | Feature Notes** panel | **Chamfer.**
- Click the chamfered edge of the slot in the detailed view
- Click the adjacent Edge.
- Click a point in space to place the **Chamfer** note
- Right-click in the graphics window and click **Done**.

41. Drag the **Chamfer** note into a new location using the grips.



11. Challenge Exercise

42. Complete detailing the drawing using the *Dimensioning Aux-Clutch-Lever Complete.dwf* as a guide.