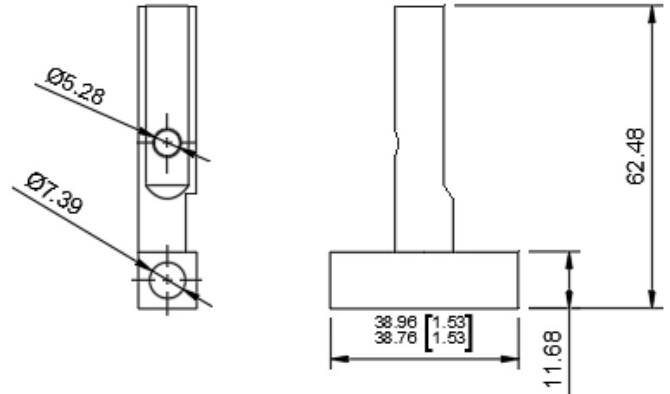# Add geometry and dimensions to a drawing

In this module, you'll add information to a drawing sheet by creating center marks, centerlines, dimensions, and notes.

Learning objectives:

- Use Centerline.
- Use Center Mark.
- Add dimensions.

The completed exercise

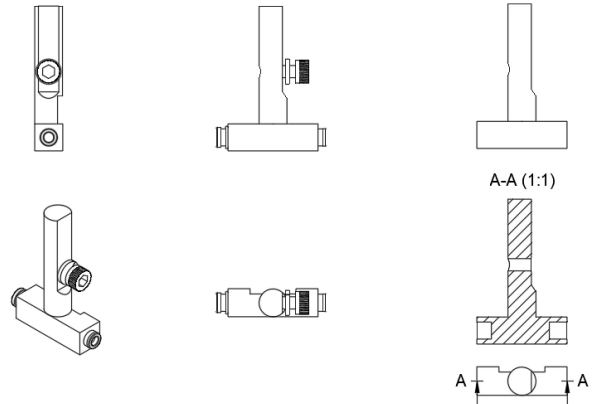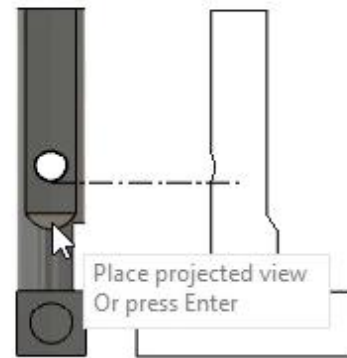**1.** Continue with the assembly and drawing files from the previous module.

Figure 1. Continue with the files from the previous module

2. Click Create> Projected View and project a view to the left of the version of the Blade Guard Post without the Fasteners.

A-A (1:1)

Figure 2. Create a projected view

3. Various types of geometry can be added to the drawing sheet; click Geometry> Center Mark.
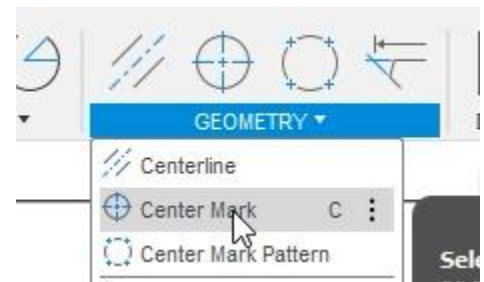

Figure 3. Open the Center Mark tool

4. Click a drawing view's circular geometry to add a center mark. This Center Mark identifies the center of the circle or arc and creates geometry that you can use to dimension the part.

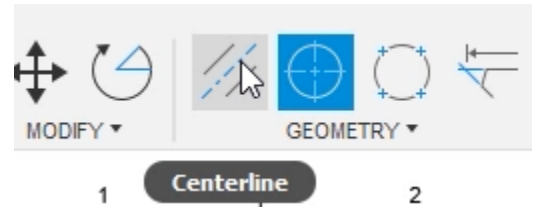*Figure 4. Add center marks to a drawing view*

5. Click Geometry > Centerline.

Figure 5. Open the Centerline tool

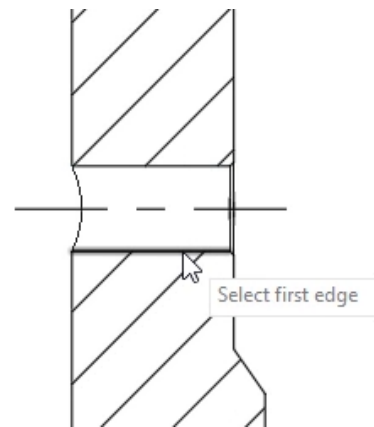**6.** Select the view's two parallel edges to create a centerline between them.

Figure 6. Create a centerline

7. Dimensions can also be added to drawing view. Expand the Dimensions group's menu and notice the many different types of dimensions you can create. Click Dimensions> Dimension.
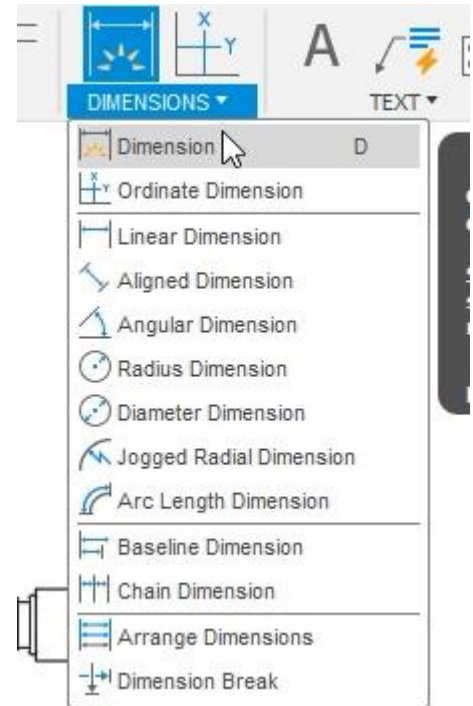

Figure 7. Open the Dimension tool

8. Select a drawing view's two parallel edges to create a dimension between them. Click a third time to place the dimension onto the drawing sheet and the location you want.
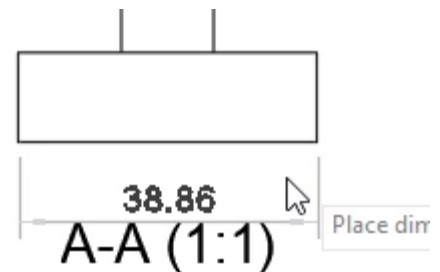

Figure 8. Create a dimension

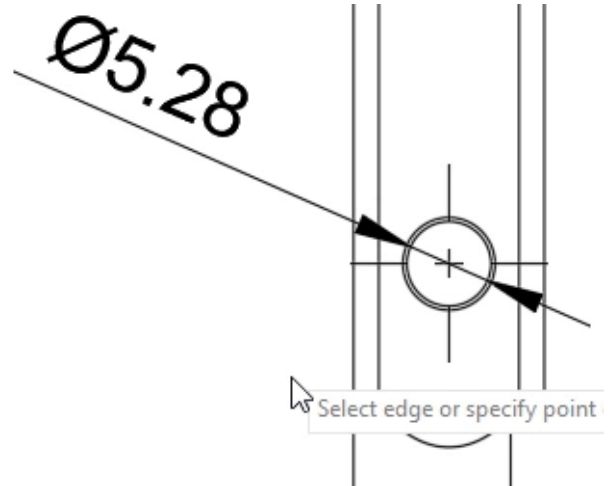**9.** Selecting a circle or an arc will create a diameter dimension.

Figure 9. Create a diameter dimension
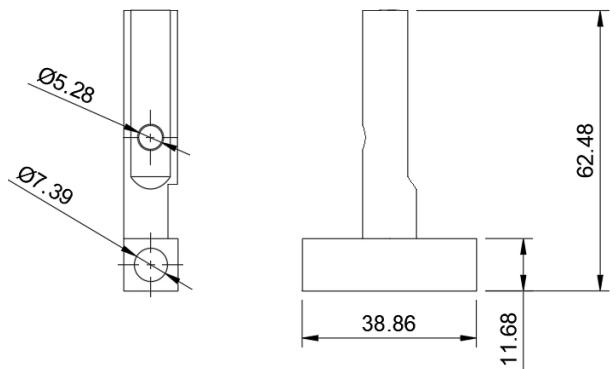
10. Continue to dimension the drawing view.

Figure 10. Create additional dimensions

11. Double-click one of the dimensions you created to open the Dimension dialog.
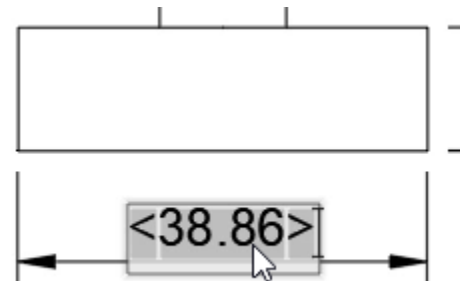
Figure 11. Double-click a dimension

12. Activate the Alternate Units and Tolerances options and note how you can modify the existing dimension. Close the dialog after you finish exploring the options.
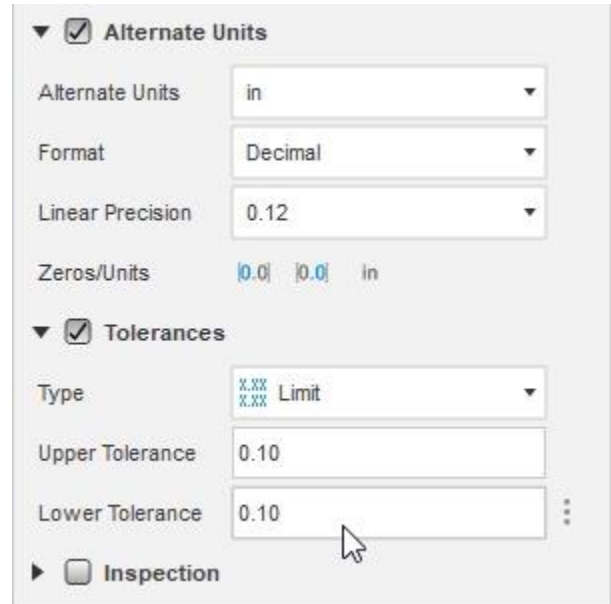

Figure 12. Inspect the dialog's options

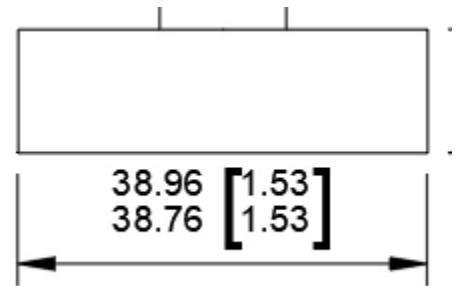**13.** The dimension you selected is updated to reflect the changes you made in the dialog.

Figure 13. Inspect the result

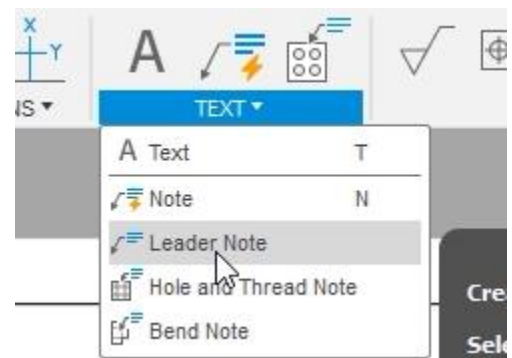**14.** Click Text> Leader Note.

Figure 14. Open the Leader Note tool

15. Click a drawing view to attach the leader, click a second time to place the note's location, then type the note you want to see. Press Enter. OK the dialog to accept the new note, save the file, then continue to the next module.
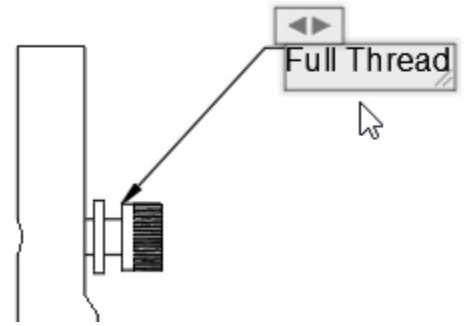

Figure 15. Create a note