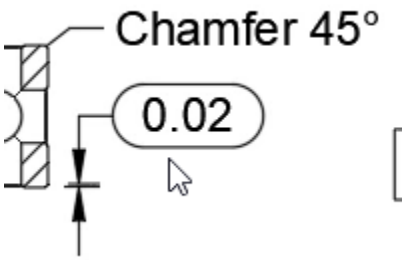
AUTODESK

Review dimension types

In this module, you'll inspect the drawing and notice the various types of dimensions it uses.

Learning objectives:

- Identify dimension types.
- Interpret dimensional tolerances.



The completed exercise

1. Continue with the *123 Block Drawing* file from the previous module.

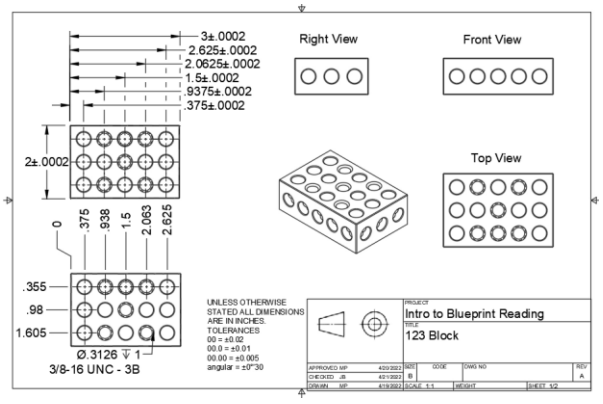


Figure 1. Continue with the file from the previous module

2. On the drawing's first sheet, notice that one of the views has two different types of dimensions. The dimensions shown in the image on the right describe the part's height and its tolerance.

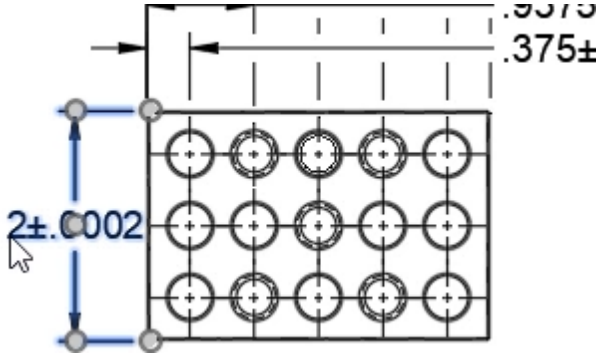


Figure 2. Inspect the view's dimension

3. The view's positional values are shown in the image on right. These dimensions describe each hole's position in relation to the part's referenced edge.

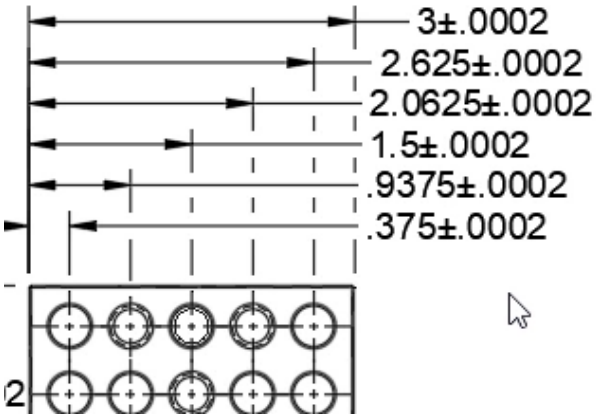


Figure 3. Inspect the view's positional values

4. Notice the note with a leader pointing to the part's geometry; this note indicates that a 3/8-16 UNC class 3B tap should be used to create this geometry. The information above the tap indicates that the hole should be predrilled to 0.3126 inches and should plunge 1 inch deep.

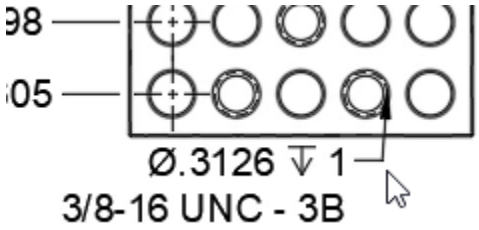


Figure 4. Notice the tap reference

5. Continue to the drawing's second sheet. Notice that there is an angular dimension specifying that the chamfered edge should be 45° in relation to the other selected edge.

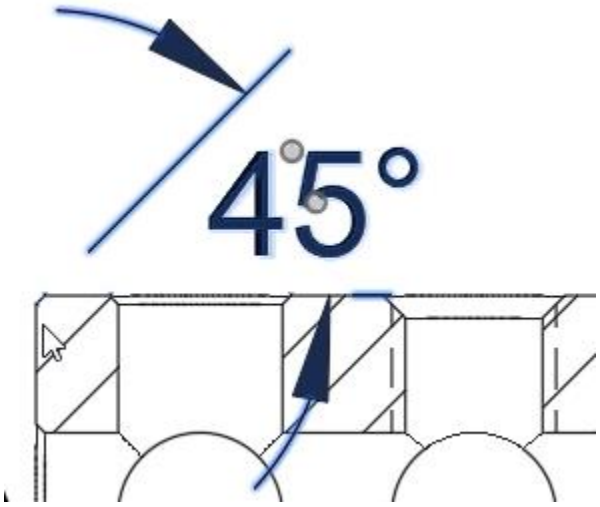


Figure 5. Notice the angular dimension

6. Also notice that the chamfer could be referenced using two other methods. A note could be applied with the chamfer's angle or a dimension could be given for the chamfer's height. The circle around the chamfer's height indicates that this geometry's dimension needs to be inspected after it has been cut. Continue to the next module.

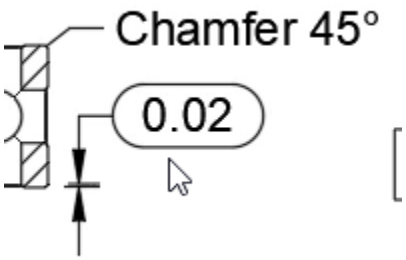


Figure 6. Inspect the other ways to reference a chamfer's geometry