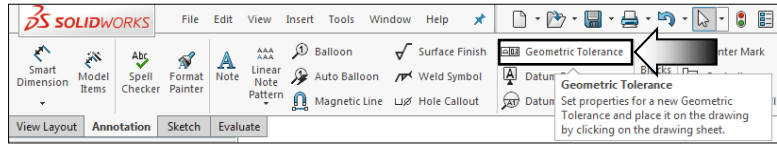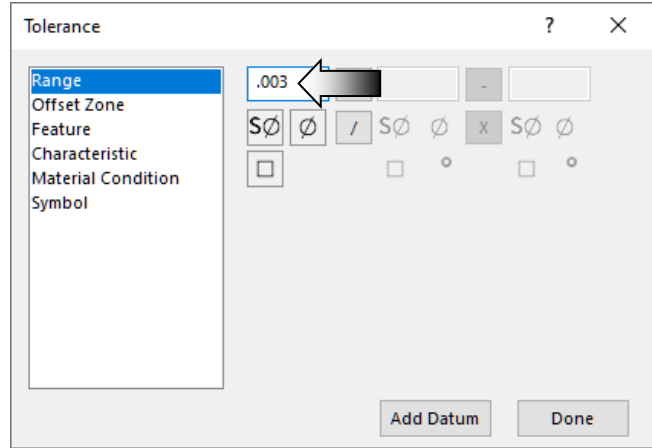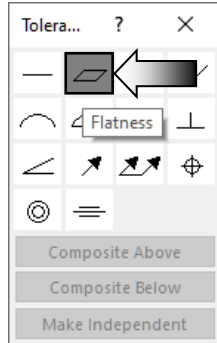## 14. Adding a Flatness tolerance:

The geometric tolerance symbol adds geometric tolerances to parts and drawings using feature control frames. The SOLIDWORKS software supports the ASME Y14.5-2009 Geometric and True Position Tolerancing guidelines.
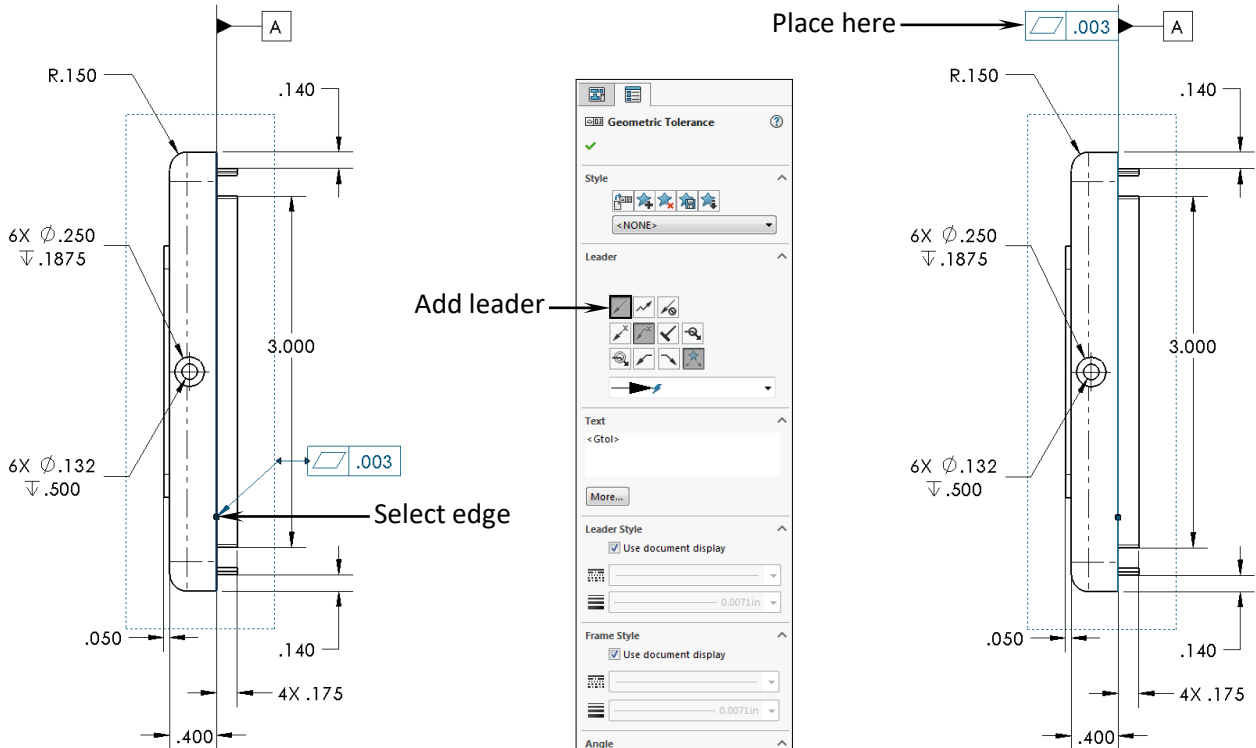
Click the **Geometric Tolerance** command.

Select the vertical edge as noted and then move upward to align it next to Datum A.

Select **Flatness** from the list.

Enter **.003** for Tolerance. Click **Done** and **OK**.

Place here

Add leader

Select edge

15. Adding a Perpendicular tolerance:

Perpendicularity in GD&T can mean two different things depending which reference feature is called out.

The normal form or **Surface Perpendicularity** is a tolerance that controls Perpendicularity between two 90° <u>surfaces</u>, or features. Surface Perpendicularity is controlled with two parallel planes acting as its tolerance zone.

Axis Perpendicularity is a tolerance that controls how perpendicular a specific <u>axis</u> needs to be to a datum. Axis Perpendicularity is controlled by a cylinder around a theoretically perfect parallel axis.
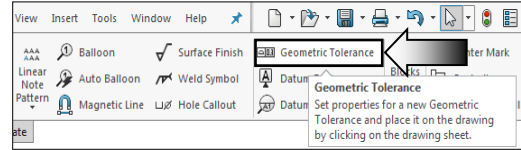
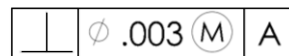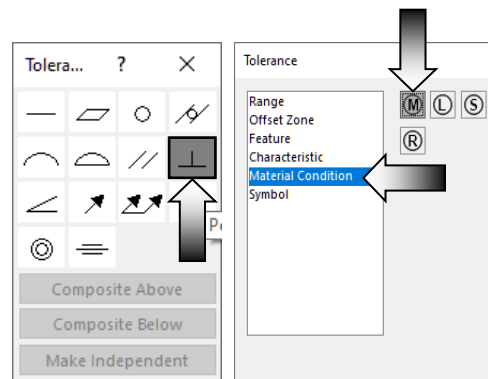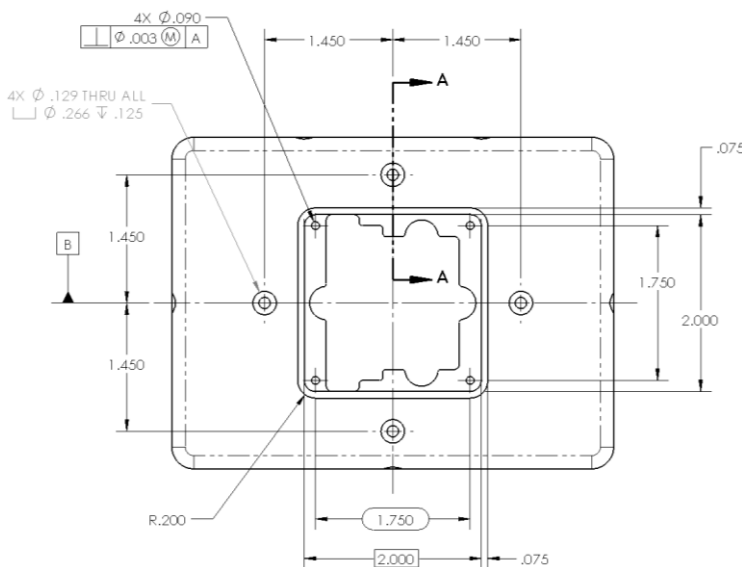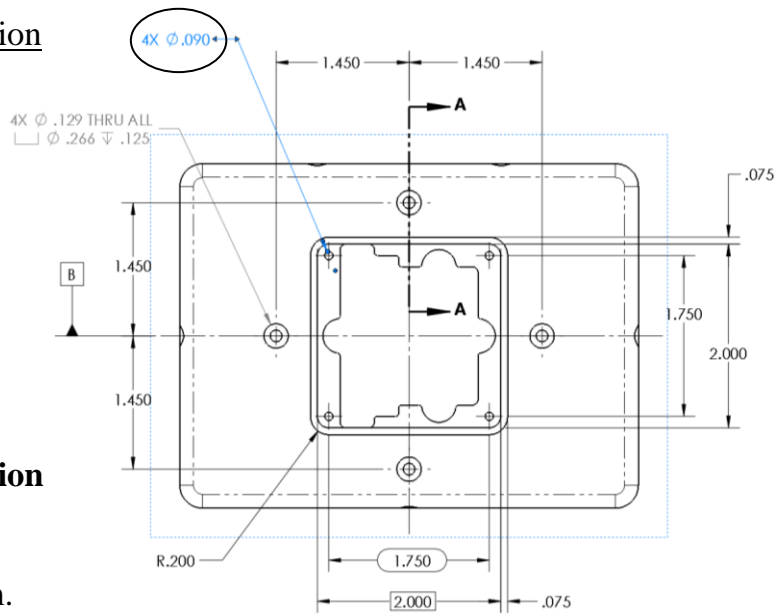Select the <u>diameter dimension</u> of the hole (circled).

Click the **Geometric Tolerance** command.

Select the **Perpendicular** symbol from the list.

Enter **Ø.003** for Tolerance.

Click the **Material Condition** and select the symbol Ⓜ.
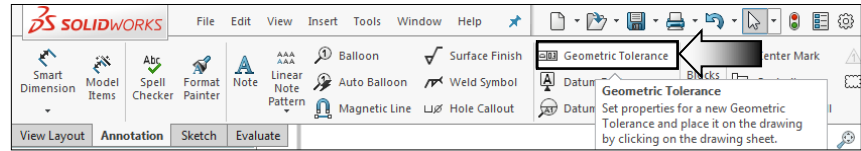
Enter **A** for Primary Datum.

The Perpendicular control frame.

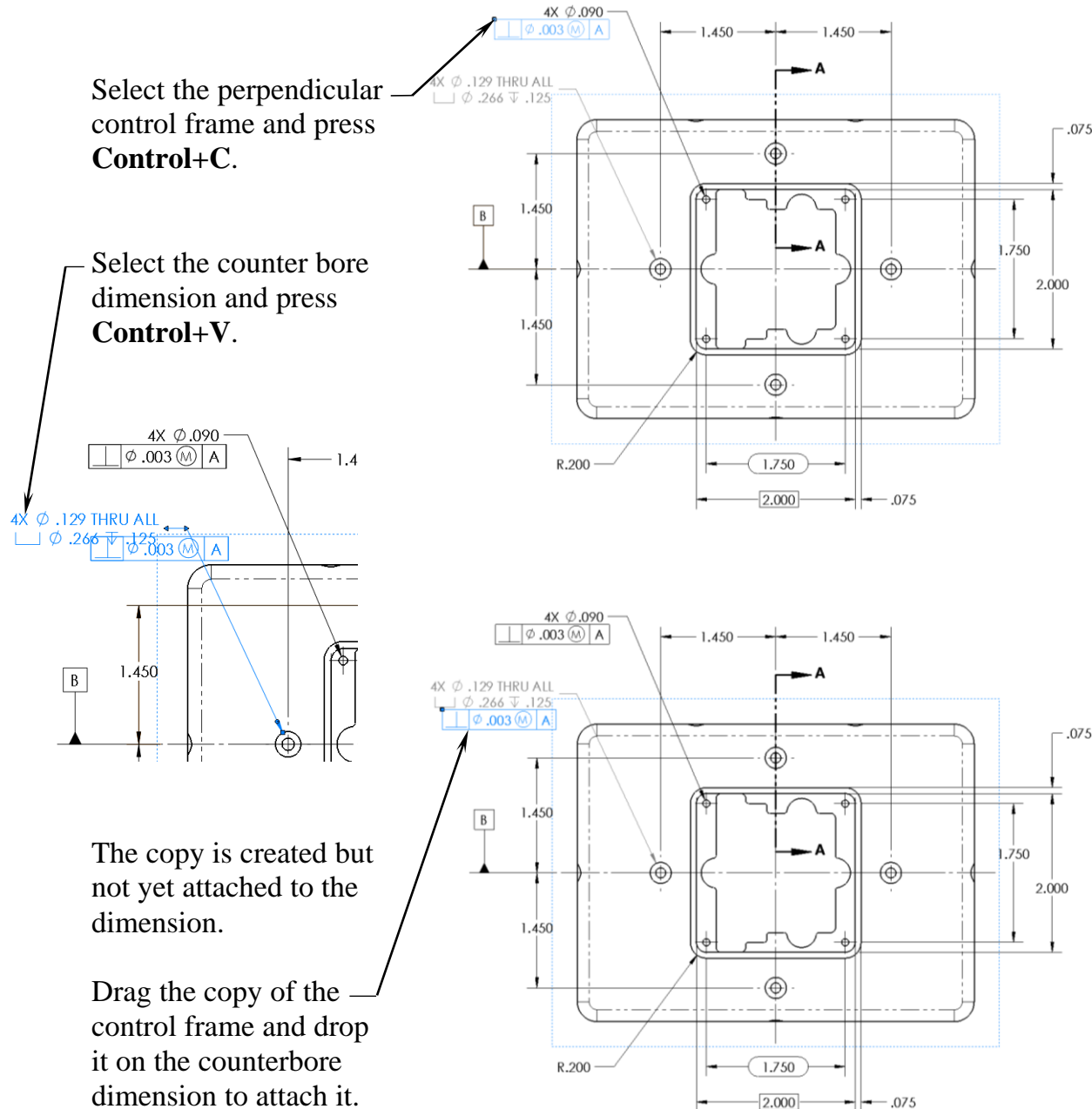Click **OK**.

16. Copying the control frame:

True position in terms of the axis, point, or plane defines how much variation a feature can have from a specified exact true location.

This step demonstrates how to make a copy of a geometric tolerance and attach it to another dimension.

Select the perpendicular control frame and press **Control+C**.

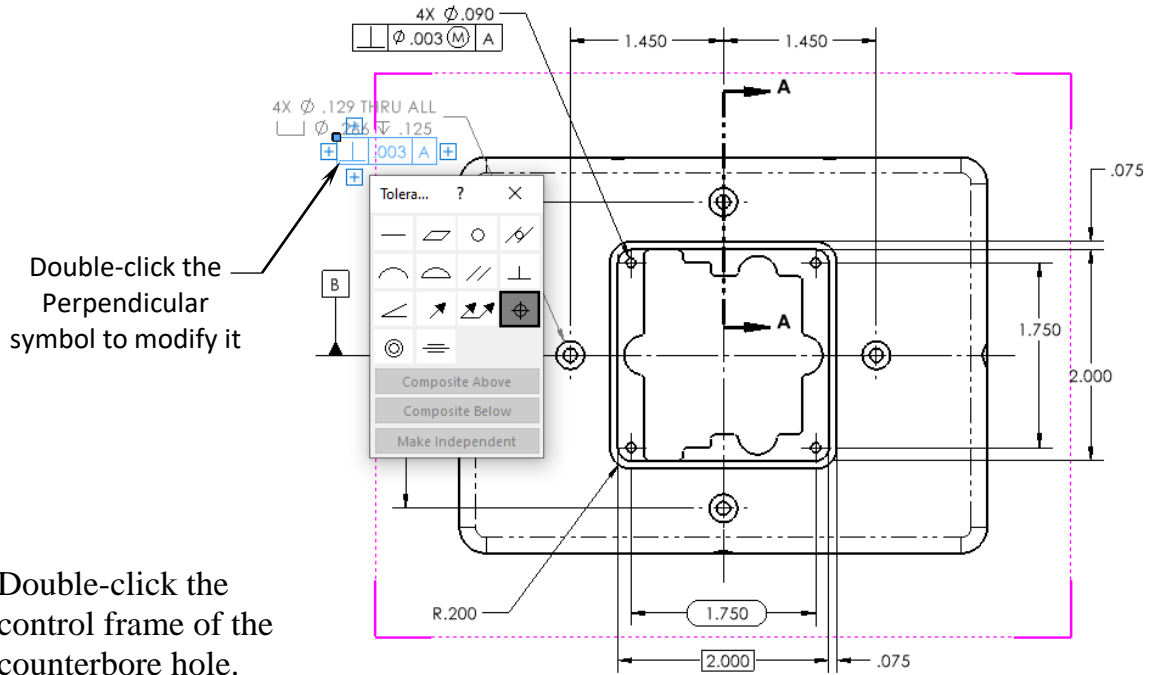Select the counter bore dimension and press **Control+V**.

The copy is created but not yet attached to the dimension.

Drag the copy of the control frame and drop it on the counterbore dimension to attach it.
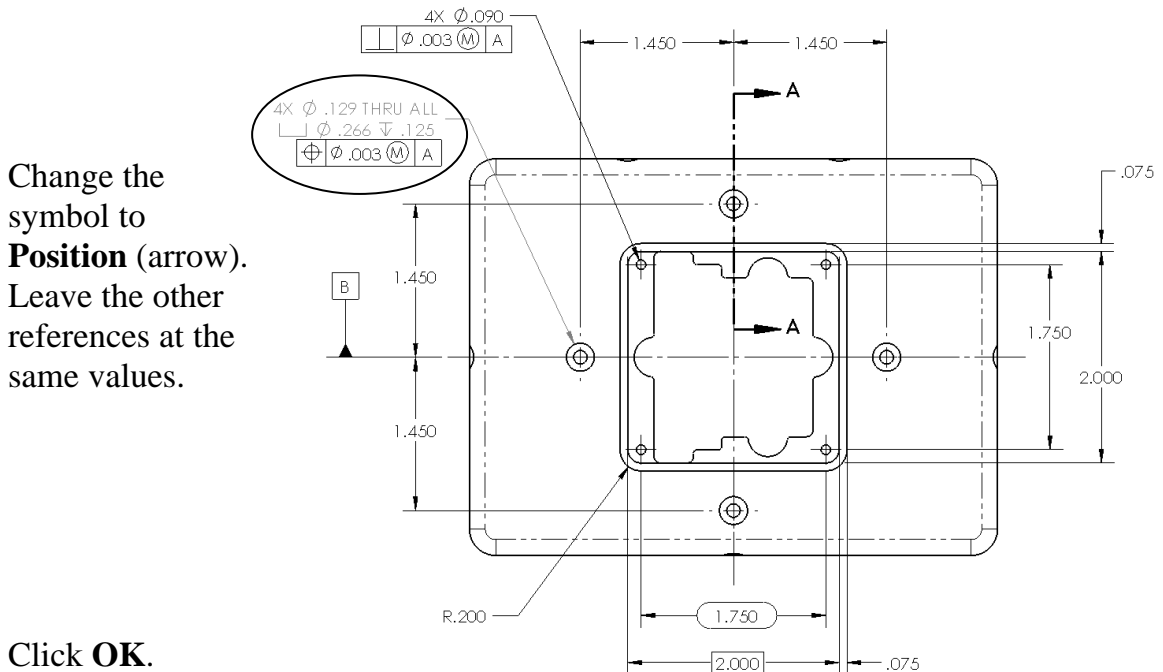
17. Modifying the geometric tolerance:

For the counter bore feature we will change the symbol to Position, or also called True Position.

True position in terms of the axis, point, or plane defines how much variation a feature can have from a specified exact true location.

Double-click the Perpendicular symbol to modify it

Double-click the control frame of the counterbore hole.

Change the symbol to **Position** (arrow). Leave the other references at the same values.
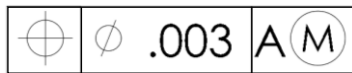
Click **OK**.

18. Adding another Position tolerance:

Zoom in on the **Back** view and locate the small hole on the lower right corner.

Select the dimension **4X Ø.125** and click the **Geometric Tolerance** command.
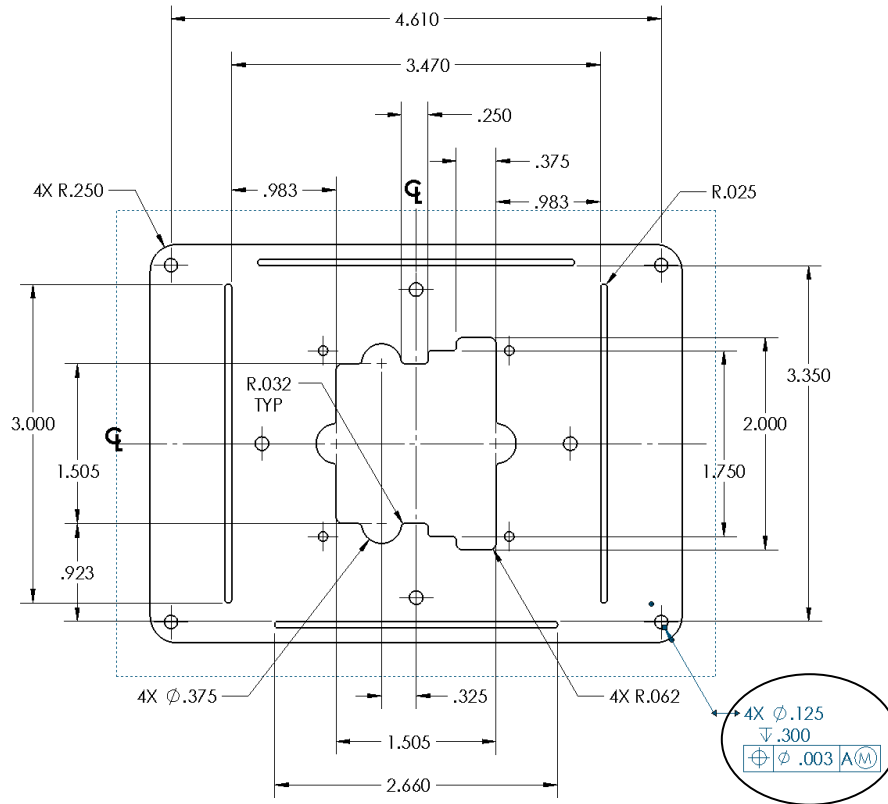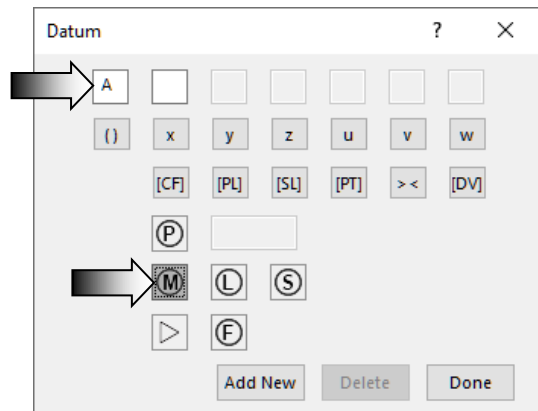

Create a True Position tolerance that looks like the one shown below.

Be sure to attach the control frame to the hole dimension (circled).

Click **OK**.

For practice purposes, we will create some other tolerances such as Symmetric, Bilateral, Limit, Basic, etc., in the next few steps.