GD&T with AutoCAD

AutoCAD allows you to add GD&T symbols to your drawings using the TOLERANCE, MLEADER, and QLEADER tools. The TOLERANCE tool displays the Geometric Tolerance dialog box, which is the primary method for adding feature control frames, geometric tolerancing, and datum target symbols. You can also connect GD&T symbols to a leader using a combination of the TOLERANCE and MLEADER tools. The MLEADER tool has replaced the QLEADER tool as the primary method for placing leaders. However, the QLEADER tool continues to provide a quick and effective option for attaching GD&T symbols to a leader.

**PROFESSIONAL TIP**

Draw GD&T symbols on a dimensioning layer so the symbols and text can plot as lines that have the same thickness as extension and dimension lines (0.01" or 0.3 mm). The suggested text font is romans.shx. These practices correspond with the standard ASME Y14.2M-1992, Line Conventions and Lettering.

**Using the TOLERANCE Tool**

The TOLERANCE tool provides options for creating feature control frames and datum target symbols using the Geometric Tolerance dialog box. See Figure 20A-1. Areas divide the Geometric Tolerance dialog box into groups of compartments that relate to the components found in a feature control frame. Each area contains two levels to define a feature control frame.

---

**feature control frame:** The rectangular frame that contains the geometric characteristic, geometric tolerance, material condition, and datum reference (if any) for an individual feature.

**geometric tolerancing:** A general term that refers to tolerances used to control the form, profile, orientation, runout, and location of features on an object.

**datum target:** A specific point, line, or area used to establish a datum.

---

Copyright by Goodheart-Willcox Co., Inc. GD&T with AutoCAD, page 1
The upper row allows you to make a single feature control frame. The lower level allows you to create a double feature control frame, as described later in this supplement. The dialog box also provides options for displaying a diameter symbol and a modifying symbol. In addition, the Geometric Tolerance dialog box allows you to display a projected tolerance zone symbol and value and part of the datum feature symbol. Projected tolerance zones are described later in this supplement.

Selecting a Geometric Characteristic Symbol

You can access geometric characteristic symbols from the Sym area located at the far left of the Geometric Tolerance dialog box. This area has two boxes that allow you to display one or two geometric characteristic symbols. Pick one of the

---

**modifying symbol**: A symbol used to establish the relationship between the size of a feature and its given dimensional and geometric tolerance.

**datum feature symbol**: Symbols used to identify datums in a feature control frame.

**geometric characteristic symbols**: Symbols that indicate specific controls related to the form of an object, orientation of features, outlines of features, relationship of features to an axis, or location of features.
Use the **Symbol** dialog box to select a geometric characteristic symbol for use in a feature control frame. Boxes in the **Sym** area to display the **Symbol** dialog box. See **Figure 20A-2**. Pick a symbol to add the symbol to the selected **Sym** box. After you make a selection, the **Geometric Tolerance** dialog box returns. Pick the same box again to select a different symbol if necessary. To remove a symbol, access the **Symbol** dialog box and pick the blank image in the lower-right corner.

**Tolerance 1 Area**

The **Tolerance 1** area allows you to enter the first geometric tolerance value applied to the feature control frame. If you are drawing a single feature control frame, enter the desired value in the upper text box. If you are drawing a double feature control frame, also enter a value in the lower text box. You can add a diameter symbol by picking the box to the left of the text box. Pick the diameter box again to remove the diameter symbol.

The box to the right of the text box adds a material condition symbol. Pick the box to display the **Material Condition** dialog box shown in **Figure 20A-3**. Pick the desired symbol to display it in the box you selected. You can choose from the maximum material condition (MMC) and least material condition (LMC) symbols. ANSI Y14.5M-1982 uses the regardless of feature size (RFS) symbol.

---

**geometric tolerance**: A tolerance used to control the form, profile, orientation, runout, and location of features on an object.

**material condition symbols**: Symbols used to modify the geometric tolerance in relation to the produced size or location of the feature.

**maximum material condition (MMC)**: The maximum allowable produced size.

**least material condition (LMC)**: The minimum allowable produced size.

**regardless of feature size** (**RFS**): A material condition in which the geometric tolerances remain the same, regardless of the actual produced size.
symbol, but ASME Y14.5M-1994 does not, because RFS is assumed unless otherwise specified. To remove a material condition symbol, pick the blank tile in the Material Condition dialog box.

In Figure 20A-4, the Sym box displays a *position symbol* and a 0.5 tolerance value in the upper text box in the Tolerance 1 area. A diameter symbol precedes the tolerance value, and the MMC symbol follows. Remember that a zero precedes metric decimals, but not inch decimals.

**Tolerance 2 Area**

The Tolerance 2 area allows you to add a second geometric tolerance to the feature control frame. This is not a common application, but is appropriate in some cases when restrictions are needed on the geometric tolerance specified.

**Figure 20A-4.**
The Geometric Tolerance dialog box with a diameter symbol, geometric tolerance value, and maximum material condition (MMC) symbol added to the Tolerance 1 area.
in the first compartment. For example, a second geometric tolerance value of 0.8 MAX maintains the specification given in the first compartment, but indicates that it cannot exceed 0.8.

**Datum Areas**

The **Datum 1** area establishes the information needed for the primary datum reference compartment. Like the Tolerance areas, this area offers two levels of text boxes to create single or double feature control frames. You can also specify a material condition symbol for the datum reference by picking the box to the right of the corresponding text box to open the Material Condition dialog box. The **Datum 2** and **Datum 3** areas allow you to specify the secondary and tertiary datum reference information. **Figure 20A-5** shows how the datum reference and related material condition symbols appear in the feature control frame. Use the **Datum Identifier:** text box to enter a datum-identifying reference letter as an element of the datum feature symbol. Use an uppercase letter.

**Projected Tolerance Zone Box and Height Text Box**

You can pick the **Projected Tolerance Zone:** box to display a projected tolerance zone symbol in the feature control frame. The **Height:** text box specifies the height of a projected tolerance zone. The projected tolerance zone symbol and the height value work together when you apply a projected tolerance zone to the drawing.

**Completing the Feature Control Frame**

After you enter the required information in the Geometric Tolerance dialog box, pick the **OK** button and pick a point to place the tolerance in the drawing. **Figure 20A-6** shows the feature control frame for the given example.

---

**Primary datum reference:** The first datum in the precedence of datums.

**Datum:** A theoretically perfect surface, plane, point, or axis.

**Secondary datum reference:** The second in the precedence of datums

**Tertiary datum reference:** The third in the precedence of datums

Copyright by Goodheart-Willcox Co., Inc. GD&T with AutoCAD, page 5
Figure 20A-5.
The order of elements in a feature control frame. (h = text height)

Figure 20A-6.
This example shows, primary, secondary, and tertiary datum reference values added and highlighted, along with the geometric tolerance value. The resulting feature control frame appears below.
NOTE
The height of the feature control frame is automatically set to twice the height of the text. Text on engineering drawings is generally .12” (3 mm) tall, which makes the feature control frame height .24” (6 mm). This complies with the ASME Y14.5M standard.

Activity
1. Start AutoCAD if it is not already started.
2. Start a new drawing from scratch or use a decimal unit template of your choice.
3. Set up the appropriate layers, including a layer for dimensions.
4. Draw the feature control frames shown in Figure 20A-7.
5. Save the drawing as ACT20A-1.

Attaching Feature Control Frames to Leaders
In many cases, a feature control frame connects to a leader. The QLEADER tool allows you to draw leader lines and access the Geometric Tolerance dialog box used to create feature control frames in one operation. This is the most effective technique for creating a feature control frame that is automatically attached and associated with a leader. You can create other GD&T symbols, such as datum feature symbols, more effectively using different methods.

Using the MLEADER Tool
You can use the MLEADER tool to create leaders, but it does not have an option to create a feature control frame at the same time. You must draw the leader separately using the MLEADER tool and add the feature control frame using the TOLERANCE tool. Apply the None multileader content type when using this method. You can draw the leader before or after the symbol. See Figure 20A-8.
Using the QLEADER Tool

The **QLEADER** tool allows you to place a leader and attach a feature control frame in one operation. Dimension style settings control some of the leader line characteristics, such as:

- **Figure 20A-7.**
  
  - **0.1**
  
  - φ 0.15 M
  
  - φ 0.08 A
  
  - φ 0.05 M A B
  
  - φ 0.08 M A B C
  
  - 0.1 A–B
Figure 20A-8.
Use the MLEADER tool to create a leader before drawing the feature control frame using the TOLERANCE tool, or add the leader to an existing feature control frame.

Pick to locate the feature control frame when prompted to enter the tolerance location

First, use the leader Landing first option to pick the location of the leader shoulder

Existing leader drawn using the None multileader content type

Second, pick the start point of the leader line

Leader added to existing feature control frame using the None multileader content type

the arrowhead size. The Settings option of the QLEADER tool controls other elements, such as the leader format and annotation style.

When you enter the QLEADER tool, use the Settings option to display the Leader Settings dialog box, shown in Figure 20A-9. Select the Annotation tab, and then pick the Tolerance radio button to display the Geometric Tolerance dialog box for creation of a feature control frame with the leader line.

Next, select the Leader Line & Arrow tab of the Leader Settings dialog box. Pick the Straight radio button to create a leader with straight-line segments. When adding a feature control frame to a leader line, you should set the maximum number of vertices in the Maximum text box of the Number of Points area to 2. When you set the maximum number of leader points to 2, you select the start and endpoints of the leader line. Then the QLEADER tool stops drawing the leader, automatically places the leader shoulder, and displays the Geometric Tolerance dialog box.

The Arrowhead area of the Leader Line & Arrow tab uses the default value assigned to leaders within the current dimension style. To change the appearance of the arrowhead, pick the drop-down list and select a terminator from the full range of choices.
You can restrict the first two segments of the leader line to certain angles using options in the Angle Constraints area of the Leader Line & Arrow tab. The options for each segment are Any angle, Horizontal, 90°, 45°, 30°, and 15°. The Ortho mode setting overrides the angle constraints, so it is advisable to turn Ortho mode off while using this tool.

Pick the OK button to exit the Leader Settings dialog box. When asked to specify the first leader point, pick the location where the arrowhead points. Then pick the end of the leader line. If the maximum number of leader points is set to 2, the Geometric Tolerance dialog box displays. Otherwise, press [Enter] to end the leader line and display the Geometric Tolerance dialog box. Specify the settings and values for the feature control frame and pick the OK button. The feature control frame connects to the leader line, as shown in Figure 20A-10.

NOTE
You can also use the LEADER tool to attach GD&T symbols to leaders automatically. However, this tool does not provide the same convenience and ability to comply with drafting standards as the QLEADER tool.
**Figure 20A-10.**
When you complete the QLEADER tool, the feature control frame connects to the leader line.

**Activity**

1. Start AutoCAD if it is not already started.
2. Start a new drawing from scratch or use a decimal unit template of your choice.
3. Set up the appropriate layers, including a layer for dimensions.
4. Use the QLEADER tool to draw the feature control frame shown in Figure 20A-10.
5. Save the drawing as ACT20A-2.

**Drawing a Projected Tolerance Zone**

AutoCAD specifies *projected tolerance zones* according to the 1982 standard. When following this standard, enter the desired geometric tolerance, diameter symbol, material condition symbol, and datum reference in the Geometric Tolerance dialog box, as previously described. Pick the Projected Tolerance Zone: box to display the projected tolerance zone symbol and enter the height in the **Height:** text box. See Figure 20A-11. Place the feature control frame in the desired location in the drawing. Notice that AutoCAD displays the projected tolerance zone height in a separate compartment below the feature control frame, in accordance with ANSI Y14.5M-1982.

*projected tolerance zone:* A tolerance zone established at true position that projects a specified distance away from the primary datum.
To specify a projected tolerance zone according to ANSI Y14.5M-1994, create a feature control frame with any modifier letters and the letter P after the tolerance value. Type the height of the projected tolerance zone after the P, and leave one space between each letter and the height value. See Figure 20A-12. Then use the CIRCLE tool to draw a circle around the modifier and the letter P. You can use the BLOCK tool, described in Chapters 25 through 28, to group the feature control frame and circles so they are selectable as a single object.

Figure 20A-11.
To add projected tolerance zone specifications in accordance with ASME Y14.5M-1982, enter the projected tolerance zone height and symbol in the Geometric Tolerance dialog box.

Activity

1. Start AutoCAD if it is not already started.
2. Start a new drawing from scratch or use a decimal unit template of your choice.
3. Set up the appropriate layers, including a layer for dimensions.
4. Draw the feature control frame and projected tolerance zone compartment shown in Figure 20A-11.
5. Save the drawing as ACT20A-3.
Several GD&T applications require that you double the feature control frame in height and provide two sets of geometric tolerancing values. These applications include **unit straightness**, **flatness**, **composite positional tolerance**, and **coaxial positional tolerance**. To draw a double feature control frame, use the **TOLERANCE** tool to create the first level of the feature control frame in the **Geometric Tolerance** dialog box as previously described. You can also use the **QLEADER** tool if you are connecting the feature control frame to a leader line. Pick the lower box in the **Sym** area. When the **Symbol** dialog box appears again,

### Figure 20A-12.
Specifying a projected tolerance zone in accordance with ASME Y14.5M-1994.

**Drawing a Double Feature Control Frame**

**unit straightness**: A geometric tolerance for material straightness given per unit length, with a separate tolerance over the total length.

**flatness**: A geometric tolerance between two parallel planes within which the surface must lie.

**composite positional tolerance**: A geometric tolerance that allows the location of a pattern of features to vary more than the tolerance of the individual features in the pattern.

**coaxial positional tolerance**: A positional tolerance controlling the axes of coaxial features (features having a common axis).
pick another geometric characteristic symbol. This results in two symbols in the **Sym** area. Continue specifying the needed information in the lower-level **Tolerance** and **Datum** compartments. See Figure 20A-13.

A *composite frame* forms when the symbols in the two **Sym** boxes are the same. However, some situations require the same geometric characteristic symbol twice, one in the upper frame and another in the lower frame. To create two single-segment feature control frames, draw two separate feature control frames and block them. If you are drawing a double feature control frame with different geometric characteristic symbols for a combination control, the feature control frame must have two separate compartments. See Figure 20A-14.

---

**Activity**

1. Start AutoCAD if it is not already started.
2. Start a new drawing from scratch or use a decimal unit template of your choice.
3. Set up the appropriate layers, including a layer for dimensions.
4. Draw the feature control frames shown in Figure 20A-13.
5. Save the drawing as ACT20A-4.

---

**Figure 20A-13.**
Specifying information for a double feature control frame in the **Geometric Tolerance** dialog box.

---

**composite frame**: A double feature control frame in which one geometric characteristic symbol is displayed in a single compartment.
You can draw datum feature symbols using the TOLERANCE and MLEADER or QLEADER tools. Usually, however, you must use a combination of TOLERANCE and MLEADER or QLEADER tools to draw an appropriate datum feature symbol. The method used to draw a datum feature symbol depends on the feature the symbol identifies. When you use the Geometric Tolerance dialog box to specify a datum feature symbol, enter the datum reference letter in the Datum Identifier: text box. See Figure 20A-15.

Figure 20A-15.
Using the Geometric Tolerance dialog box to enter a datum-identifying reference letter. The letter creates the datum feature symbol.
Options for Drawing Datum Feature Symbols

The datum feature symbols shown in Figure 20A-16 are drawn using the TOLERANCE and MLEADER or QLEADER tools. One option is to use the TOLERANCE tool first to place the datum identifier and then add a leader that connects the feature to the identifier. The other option is to draw a leader first and then use the TOLERANCE tool to add the datum identifier. This usually requires you to move the datum identifier to the correct location using object snaps. Figure 20A-17 shows both methods.

When you use the MLEADER tool to add the leader, create a separate multi-leader style with a Datum triangle filled arrowhead symbol, set the maximum leader points to 2, do not include a landing, and use the None multileader content type. When you use the QLEADER tool to add the leader, create a dimension style that uses the Datum triangle filled leader, use the None annotation type, and set the maximum leader points to 2.

PROFESSIONAL TIP
When a datum feature symbol requires a shoulder, add the shoulder manually by picking a third point. This avoids shifting the angle of the leader line.
Adding Datum Feature Symbols to Angled Surfaces

You must follow specific steps in order to add a datum feature symbol to an angled surface, as shown in Figure 20A-18. One option is to use the QLEADER tool. Before adding the leader, create a dimension style that uses the Datum triangle filled leader. Then enter the QLEADER tool and use the Settings option to open the Leader Settings dialog box. Select the Annotation tab and pick the Tolerance radio button. Select the Leader Line & Arrow tab of the Leader Settings dialog box and pick the Straight radio button. When adding a datum feature to a leader line, you should set the maximum number of vertices in the Maximum text box of the Number of Points area to 3. This allows you to construct the leader shoulder manually. If you let AutoCAD form the leader shoulder automatically, it shifts the angle of the leader line.
Select the **OK** button to exit the **Leader Settings** dialog box. Pick the leader start point and then the next leader point. The second point must create a line segment that is perpendicular to the angled surface. Pick the third point to define the length of the leader shoulder. If the maximum number of leader points is set to 3, the **Geometric Tolerance** dialog box displays. Otherwise, press **[Enter]** to end the leader line and display the **Geometric Tolerance** dialog box. Specify a value in the **Datum identifier** text box and pick the **OK** button.

**PROFESSIONAL TIP**

Another option for placing GD&T symbols is to create your own blocks with attributes. You can insert blocks into the drawing and adjust the attribute data as needed. You can also add blocks to multileader lines using the Block multileader content type. Chapters 25 through 28 provide detailed information on how to create blocks.
Drawing Basic Dimensions

Figure 20A-19 shows a basic dimension. You can draw basic dimensions automatically by setting a basic tolerance in the Tolerances tab of the Modify Dimension Style dialog box. Typically, you establish a separate dimension style for basic dimensions because not all of the dimensions on a drawing are basic.

The height of the basic dimension rectangle is twice the height of the text, as shown in Figure 20A-20. Text on engineering drawings is generally .12” (3 mm) tall, which makes the basic dimension rectangle height .24” (6 mm). As a result, the distance from the text to the basic dimension rectangle should be equal to half the text height. For example, if the height of the drawing text is .12”, the space between the text and the basic dimension rectangle should be .06” to result in a .24” high frame. The Offset from dim line: setting in the Text tab of the New (or Modify) Dimension Style dialog box controls the distance from the text to the basic dimension rectangle. The setting also controls the gap between the dimension line and the dimension text for linear dimensions.

Figure 20A-19.
A basic dimension.

basic dimension: A theoretically perfect dimension used to describe the exact size, profile, orientation, and location of a feature.
Figure 20A-20.
The height of the rectangle drawn around basic dimension text is twice the text height by default.

![Diagram showing basic dimension with text height and tolerance rectangle](image)

The number of times or places may be applied to a basic dimension by placement inside or outside of the basic dimension symbol.

**Note**

Picking the **Draw frame around text** check box in the **Text** tab of the **New** (or **Modify**) **Dimension Style** dialog box also activates the basic tolerance method.

**Activity**

1. Start AutoCAD if it is not already started.
2. Start a new drawing from scratch or use a decimal unit template of your choice.
3. Set up the appropriate layers, including a layer for dimensions.
4. Create a new dimension style named GDT. With the dimension text height set to .12”, change the space between the text and the basic dimension rectangle to .063” (half the text height).
5. Draw the basic dimension shown in Figure 20A-19.
6. Save the drawing as ACT20A-6.
Editing Feature Control Frames

A feature control frame acts as one object. The entire object selects when you pick any location on the frame. You can edit feature control frames using editing tools such as ERASE, COPY, MOVE, ROTATE, and SCALE. The STRETCH tool only allows you to move a feature control frame. This effect is similar to the result of using the STRETCH tool with text objects.

You can edit the values inside a feature control frame using the DDEDIT tool. When you enter this tool and select a frame, the Geometric Tolerance dialog box displays with the current values. After you make changes, pick OK to update the feature control frame.

You can also use the DDEDIT tool to edit basic dimensions. Select the basic dimension to display the dimension text in the mtext editor. You can then edit the basic dimension as you would any other dimension.

NOTE

If you double-click on a dimension object, AutoCAD opens the Properties palette.