FlowCAD

Using Dimension Environment

Allegro PCB Editor

Application Note | V1

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Intro

Allegro PCB Editor offers an environment created specifically for working with dimensions, which is known as the Dimension Environment. This environment provides the user with enhanced dimensioning capabilities, which are covered in this application note.

This document is intended for users of Allegro PCB Editor who want to gain an understanding of the Dimension Environment and its capabilities.

Dimension Environment gives users access to various dimensioning features and modes for creating and editing dimensions.

Using the Dimension Environment

The Dimension Environment can be invoked by clicking **Manufacture > Dimension Environment** (see Figure 1) from the menu.
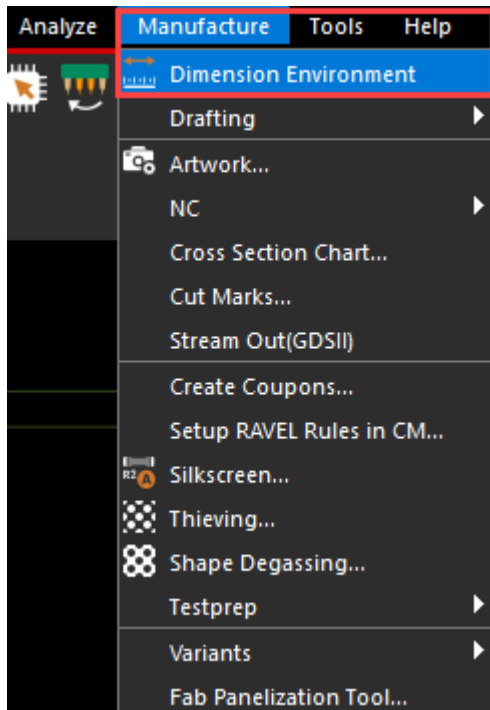
Figure 1: Invoke Dimension Environment

Dimension Environment Menu

When in the Dimension Environment, clicking the **RMB (right mouse button)** will show a pop-up menu containing relevant commands and options for dimensioning.

Done	F6
Oops	F8
Cancel	F9
Next	Ctrl+F2
Reject	
Parameters	
✓ Linear dimension	
Datum dimension	
Angular dimension	
Leader line	
Diametral leader	
Radial leader	
Balloon leader	
Chamfer leader	
Show dimensions	
Align dimensions	
Lock dimensions	
Unlock dimensions	
Z-Move dimensions	
Delete dimensions	
Refresh dimensions	
Instance parameters	
Move text	
Mirror text	
Change text	
Edit leaders	
Delete vertex	
Snap pick to	▶

Parameters

Clicking **Parameters** will bring up the **Dimensioning Parameters** dialog, where all options related to dimensioning can be specified. If a global parameter is changed, it applies to all new dimensions as well as existing dimensions.

Instance Parameters

This option allows you to change instance-specific parameters for the specified dimensions. When an instance-specific parameter is changed, it is applied only to the selected dimensions.

Dimension Mode

The currently active dimension mode is indicated by a ticked checkbox (✓) next to it.
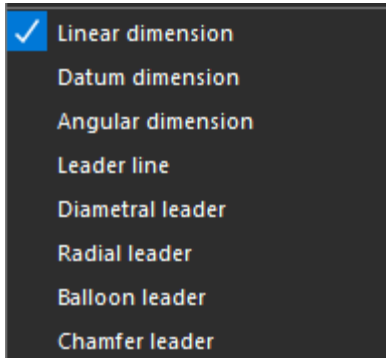
Snap Pick To

When performing dimensioning routines, the Snap pick functionality is accessible from the **Snap pick to** submenu at the bottom.

Parameters

The **Dimensioning Parameters** dialog allows you to specify various options for dimensions. The parameters are either global or instance-specific.

Dimensioning Parameters – General

The General tab (Figure 2) allows you to set the conformance standard and default measurement units to be used.
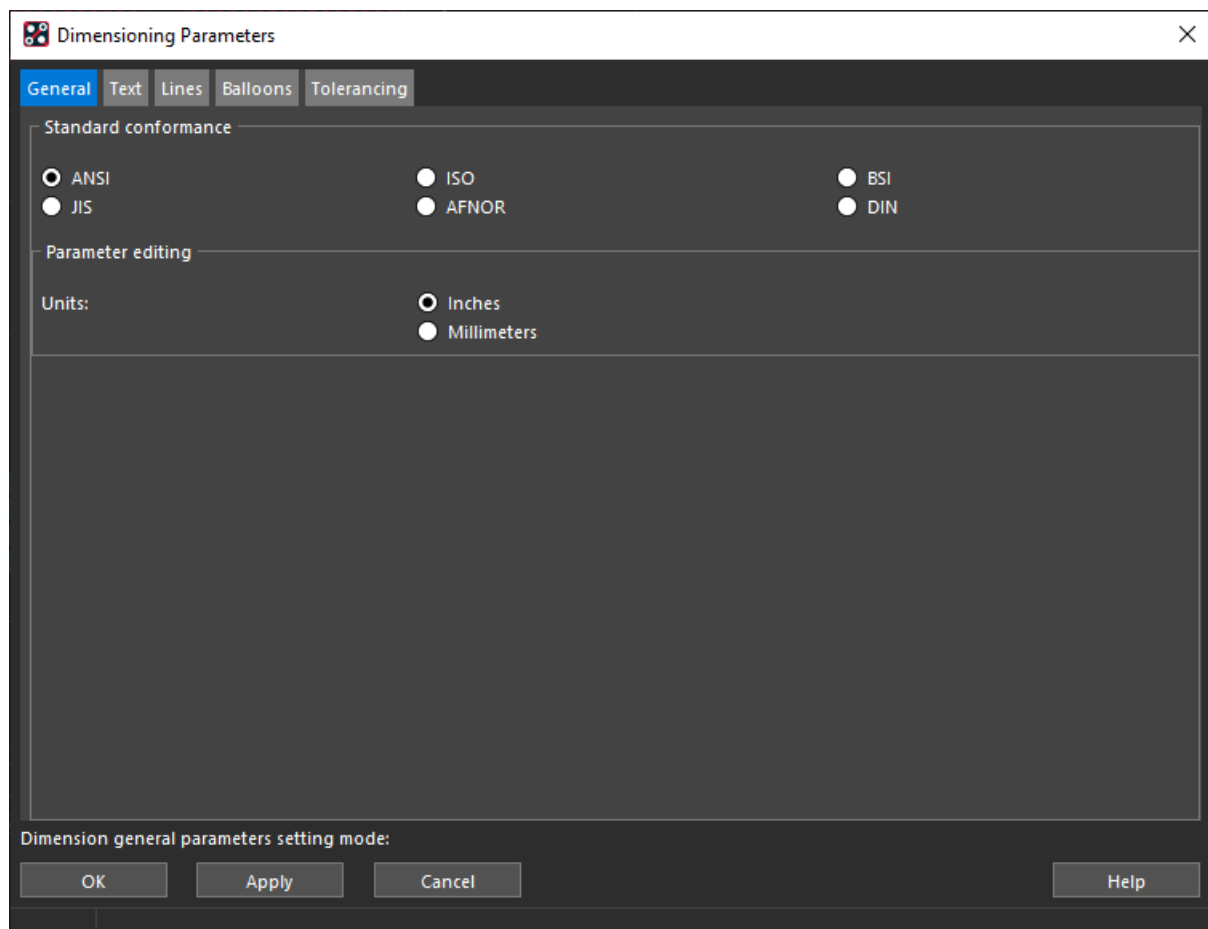
Figure 2: Dimensioning Parameters – General

Standard Conformance

The Layout Editor automatically matches the parameter settings to the standard you have chosen, to accommodate your unique dimensioning style. For example, if you specify **AFNOR**, all dimensions will be in millimeters.

The following conformance standards are supported:

- **ANSI** – American National Standards Institute (default)
- **BSI** – British Standards Institute

- **DIN** – German Industrial Normal
- **ISO** – International Organization for Standardization
- **JIS** – Japanese Industrial Standard
- **AFNOR** – French Association of Normalization

Parameter Editing

Units: This setting defines whether elements comprising dimension graphics are rendered using inches or millimeters as the unit of measurement in the **Dimension lines, Extension lines,** and **Balloons** sections in the **Dimensioning Parameters** dialog.

Note

Dimensioning units have no impact on current database units or accuracy.

Dimensioning Parameters – Text

The **Text** tab (Figure 3) controls the appearance of the text used in dimensioning designs. When dimensioning in other than current database units and precision, the dimensional values are converted from current database units to the set parameters of units and accuracy for primary and secondary dimensions.
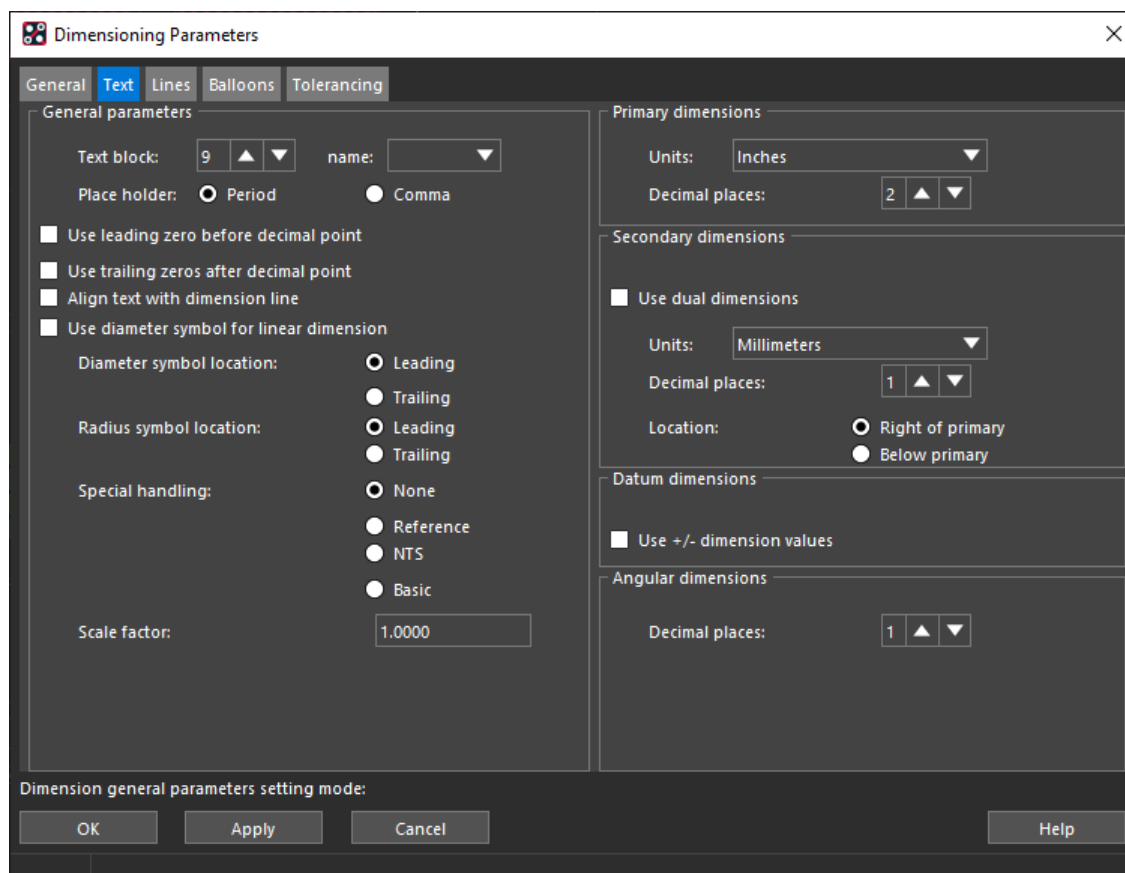

Figure 3: Dimensioning Parameters – Text

General Parameters

- **Text block** – This option specifies which predefined text block to use.
- **Name** – This option specifies the name of the text block.
- **Place holder** – This option specifies either a period (default) or comma as the placeholder.
- **Use leading zero before decimal point** – This option allows the use of a leading zero for dimensioning if the tolerance value is less than 1.
- **Use trailing zeros after decimal point** – This option allows the use of trailing zeros for dimensioning elements other than tolerance values.
- **Align text with dimension line** – This option aligns the text with the dimension line.
- **Use diameter symbol for linear dimension** – This option allows you to use the diameter symbol for linear dimension.
- **Diameter symbol location** – This option specifies the diameter symbol location as either leading or trailing.
- **Radius symbol location** – This option specifies the radius symbol location as either leading or trailing.
- **Special handling** – This option sets the following special features in the documentation of your design:
  - **Reference** – This option documents design information not used during manufacturing display. These dimensions display in parentheses.
  - **NTS** – This option applies dimensions to unscaled details. These dimensions display with underlined text.
  - **Basic** – This option is used for geometric tolerancing. These dimensions are enclosed in a box, without tolerances applied.
- **Scale factor** – This option specifies a scale factor for the text.

Primary Dimensions

- **Units** – This option controls the units of measure in which the dimension text itself displays.
- **Decimal places** – This option specifies the number of decimal places for the primary units in your design.

Secondary Dimensions

- **Use dual dimensions** – This option specifies both primary and secondary units of measure and accuracy. Primary units appear in front or on top of secondary units in your design.
  Note
  When active, dual dimensioning features are applied to all dimension types.

- **Units** – This option specifies the secondary units for your design. Secondary units display inside brackets.
- **Decimal places** – This option specifies the number of decimal places for the secondary units in your design.
- **Location** – This option specifies whether you want the secondary units to appear below or to the right of the primary units.

Datum Dimensions

- **Use + / - dimension values** – This option specifies the direction (left / right or above / below) along with the absolute distance from the reference datum dimension.
- When enabled, the 'X' relative datum to the left of the reference datum dimension and the 'Y' relative datum dimension below the reference datum dimension are labelled with a '-'. (for example: '-1.0').

Angular Dimensions

- **Decimal places** – This option specifies the number of decimal places for the angle.

  Note
  Decimal place settings for primary, secondary, and angular dimensions in the **Dimensioning Parameters** dialog do not affect the existing dimensions.

Dimensioning Parameters – Lines

The **Lines** tab (see Figure 4) controls the appearance of dimension lines and leaders that are used in dimensioning designs. You can specify the termination types and sizes for all dimensioning commands. For linear dimensions, you can choose dimension line termination types (or None). You can also choose whether arrows are to appear inside or outside of the extension lines.
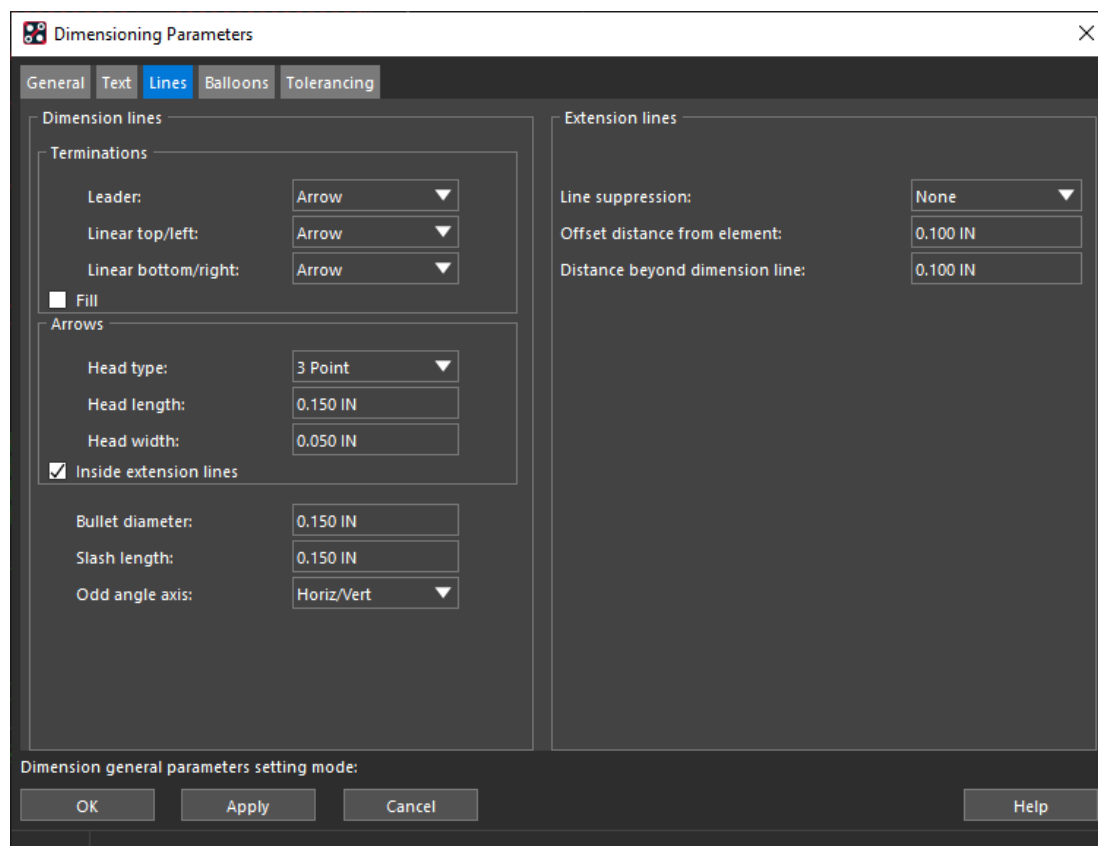

Figure 4: Dimensioning Parameters – Lines

Dimension Lines

- **Terminations**
    - **Leader** – Type of leader to use
    - **Linear top / left** – Type of leader to use for linear top / left
    - **Linear bottom / right** – Type of leader to use for linear bottom / right
- **Arrows**
    - **Head type** – Arrowhead type to use: 3-point, 4-point, or Open
    - **Head length** – Length of arrowhead
    - **Head width** – Width of arrowhead
    - **Inside extension lines** – Placement of arrows inside extension lines
- **Bullet diameter** – This option specifies the diameter of a bullet.
- **Slash length** – This option specifies the length of a slash.
- **Odd angle axis** – This option specifies the dimension for the X and Y distances or the Odd Angle point-to-point distance in cases where two linear dimension points are not on the same axis. Choices are **Horiz / Vert** and **Odd Axis**.

Extension Lines

- **Line suppression** – This option specifies which lines in the design to suppress. The choices are **None, Top / Left, Bottom / Right,** or **Both**.
- **Offset distance from element** – This option specifies the distance away from the element on which you want the extension line.
- **Distance beyond dimension line** – This option specifies the distance beyond the dimension line on which you want the extension line.

Dimensioning Parameters – Balloons

The **Balloons** tab (see Figure 5) controls the appearance of balloons that are used in documenting designs. Use this option to control the appearance of informational balloons that you use in your design.

- Balloon types (circle, square, triangle, or oblong), size, and the size of the text block, the next value, and the auto-increment value.
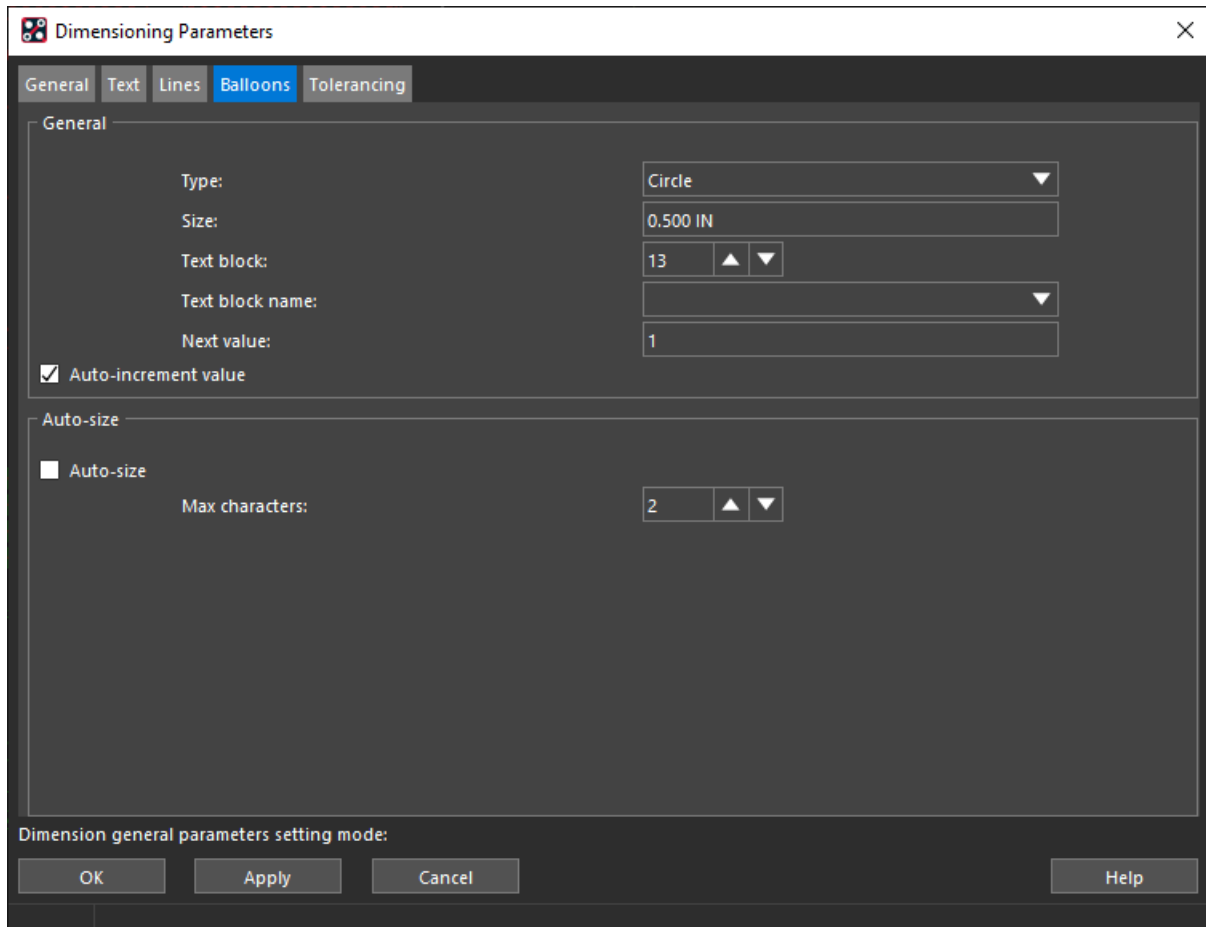- Auto-size and maximum number of characters.

Figure 5: Dimensioning Parameters – Balloons

General

- **Type** – This option specifies the balloon shape. You can choose from circle, square, triangle, or oblong.
- **Size** – This option specifies the size of the balloon.
- **Text block** – This option specifies the size of the text block.
- **Text block name** - This option specifies the name of the text block.
- **Next value** – This option specifies the value in the balloon.
- **Auto-increment value** – This option specifies the increment of the value in the balloon.
- **Auto-size** – This option defines whether or not the balloon is automatically sized.
- **Max characters** – This option specifies the maximum number of characters within the balloon.

Dimensioning Parameters – Tolerancing

Limit dimensioning, and coordinate and angular tolerancing (see Figure 6) enables you to specify the minimum and maximum amounts by which dimensions can vary during the manufacturing process.
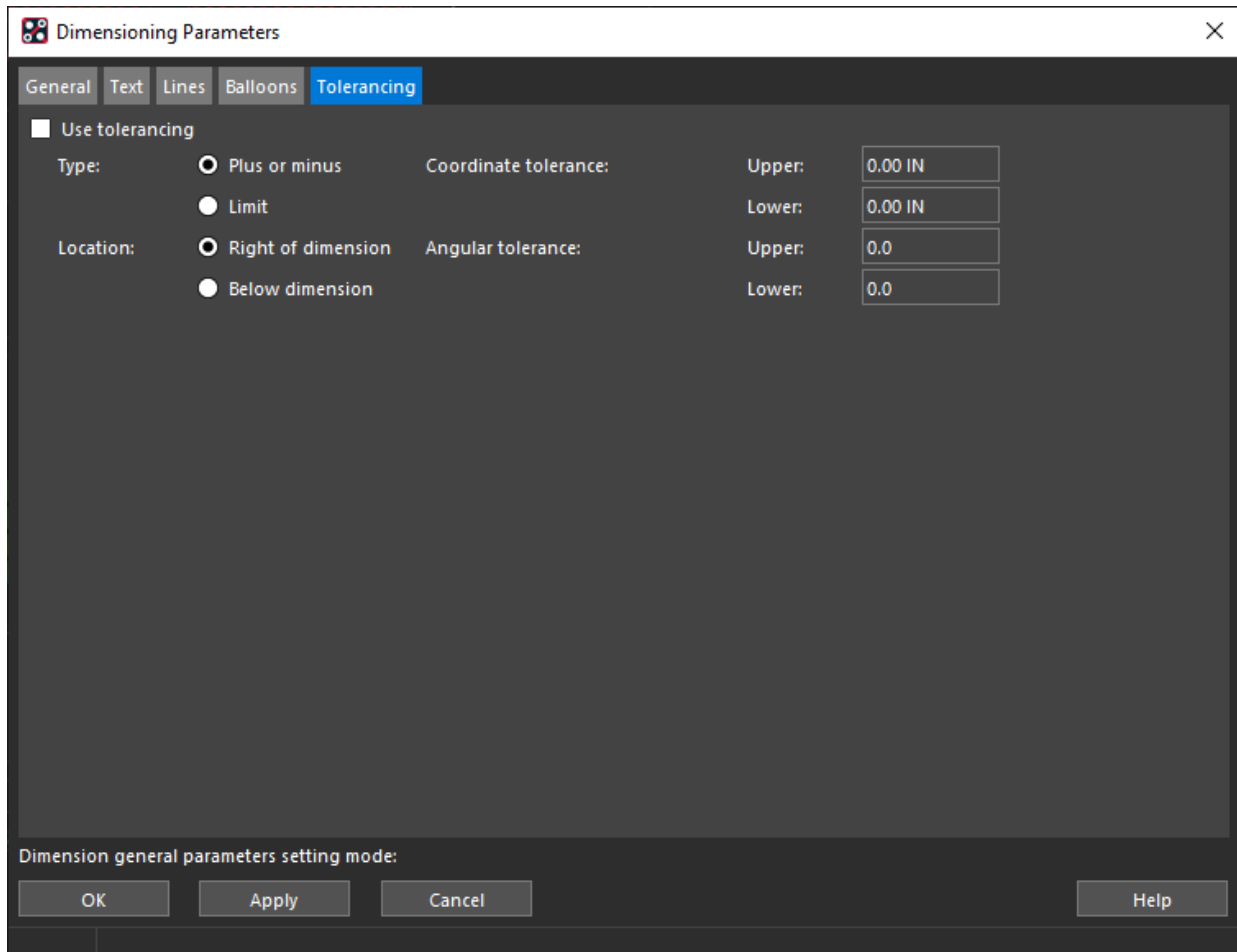
Figure 6: Dimensioning Parameters – Tolerancing

- **Use tolerancing** – This option specifies whether or not you want tolerancing to occur. When enabled, tolerances and limits are applied to all created dimensions.
- **Type** – This option specifies whether you want to set a limit or range in the tolerance. You can apply a plus (+) and / or minus (-) tolerance to dimensions.
- **Location** – This option specifies the location of the display.
- **Coordinate tolerance** – This option sets the limits for the upper and lower coordinates.
- **Angular tolerance** – This option sets the limits for angular tolerance.

Dimensioning Features

PCB Editor provides the following dimensioning features:

- Special dimensioning
- Dual dimensioning
- Coordinate tolerancing and limit dimensioning

Special Dimensioning

Special dimensioning may be required during the documentation process. The Special handling setting in the Text tab of the Dimensioning Parameters dialog (Figure 7) enables you to define these dimensions. See Figure 8 for an example of special dimensioning.
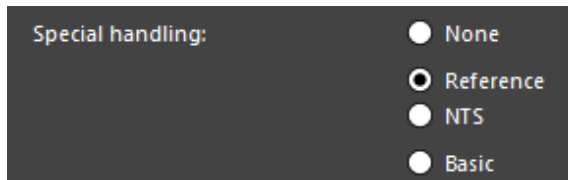

Figure 7: Special dimensioning settings

When active, special dimensioning features are applied to all dimension types.
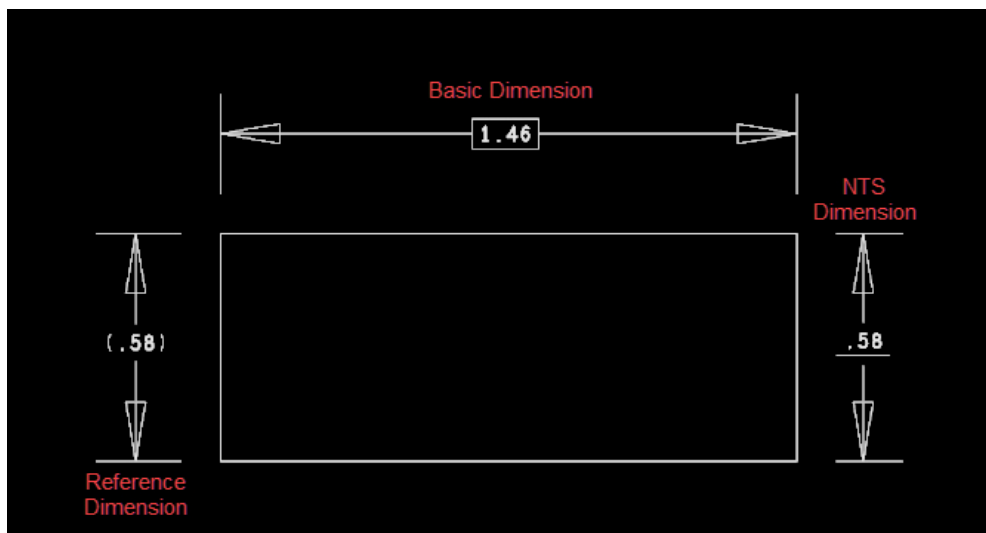

Figure 8: Special dimensioning

Dual Dimensioning

Dual dimensioning lets you specify the primary and secondary units of measure and accuracy. Primary units appear in front of or on top of secondary units, enclosed in brackets, whereas secondary units appear below or to the right of the primary dimension. See Figure 9 for an example of dual dimensioning. See the primary and secondary dimensions in the Text tab in the Dimensioning Parameters dialog (Figure 10).
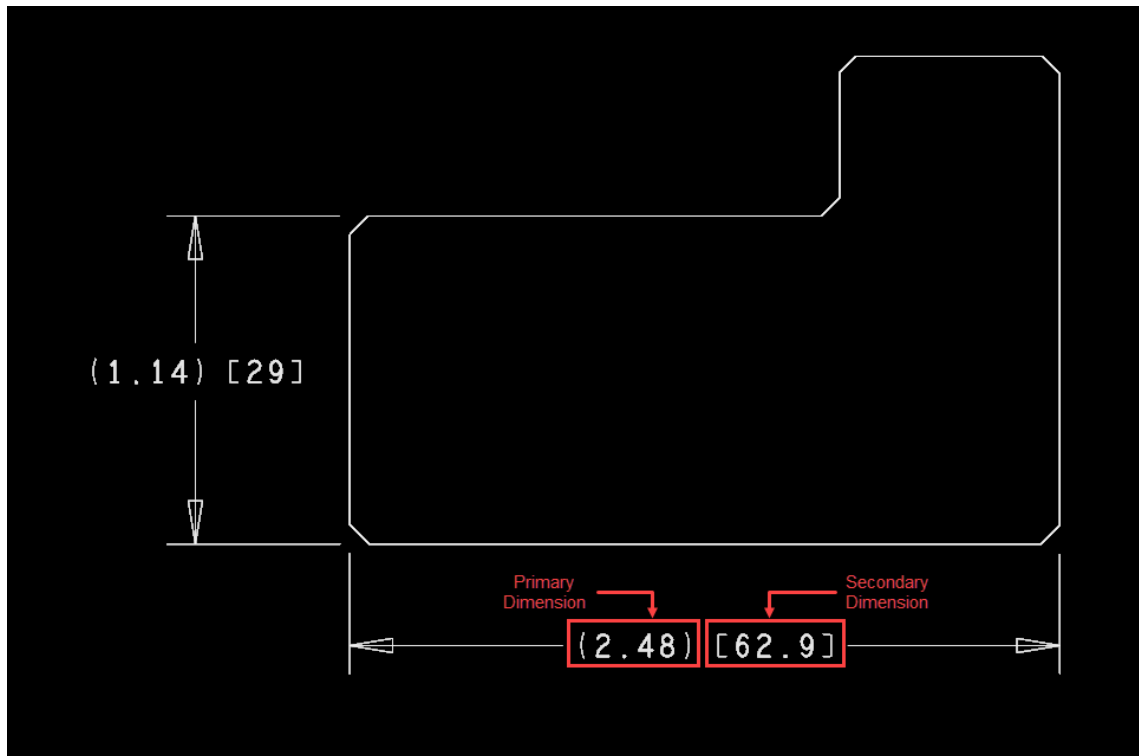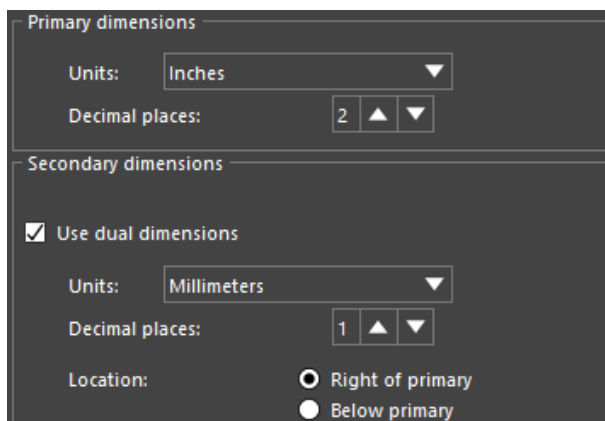
Figure 9: Dual dimensioning

Figure 10: Dual dimensioning settings

Coordinate Tolerancing and Limit Dimensioning

Limit dimensioning, and coordinate and angular tolerancing allow you to specify the minimum and maximum amounts by which dimensions can vary during the manufacturing process. You can apply a plus (+) and / or minus (-) tolerance to dimensions. These settings can be changed in the **Tolerancing** tab (Figure 11) in the **Dimensioning Parameters** dialog. Figure 12 shows an example of toleranced dimensions.
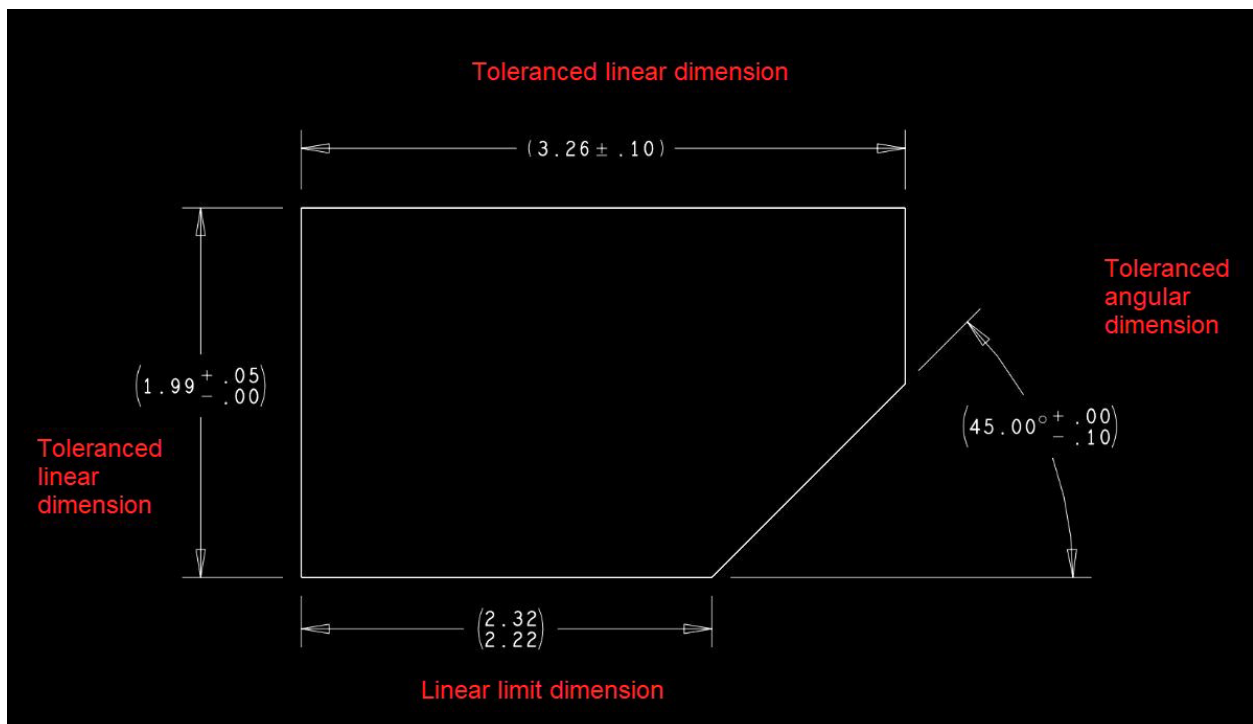
Figure 11: Dimension tolerancing settings

Figure 12: Toleranced dimensions

Dimension Associativity and Editing

With associative dimensioning, objects involved in the initial creation of the dimension continue to remain associated with the dimension symbol. This is accomplished internally in the database by putting the dimension symbol and the objects that it dimensions, in a group relation. When an object being dimensioned is either moved or deleted, an associated dimension can be implicitly moved and updated, or deleted where needed. See Figure 13 and Figure 14.
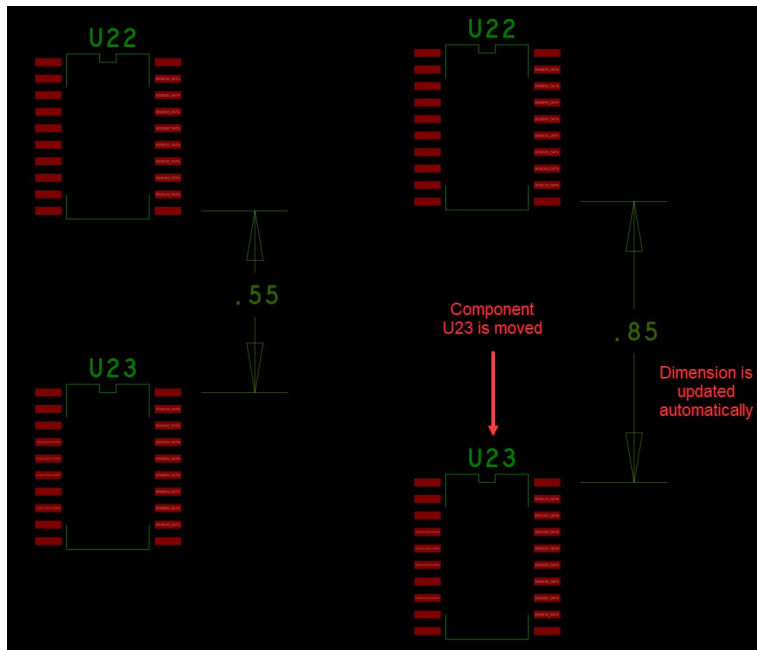

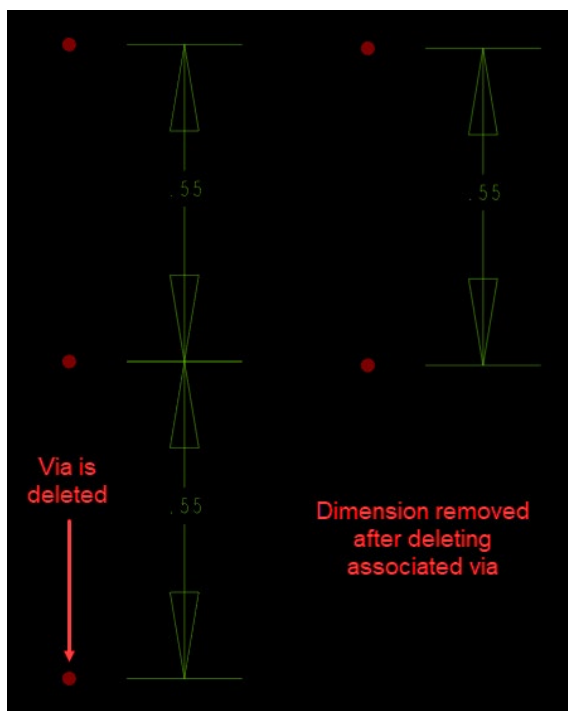Figure 13: Dimension updated after moving associated symbol

Figure 14: Dimension removed after deleting associated via

Dimensioning Modes

Dimension modes let you create and edit different types of dimensions. You can create multiple dimensions within each mode until the mode is switched to a new one or **Done** or **Cancel** is clicked, which will terminate the command.

Linear Dimension

Linear dimensioning enables you to add standard linear dimensions of objects or add linear dimensions between two user-defined points in a layout. These dimensions can be horizontal, vertical, or at an angle (see Figure 15).
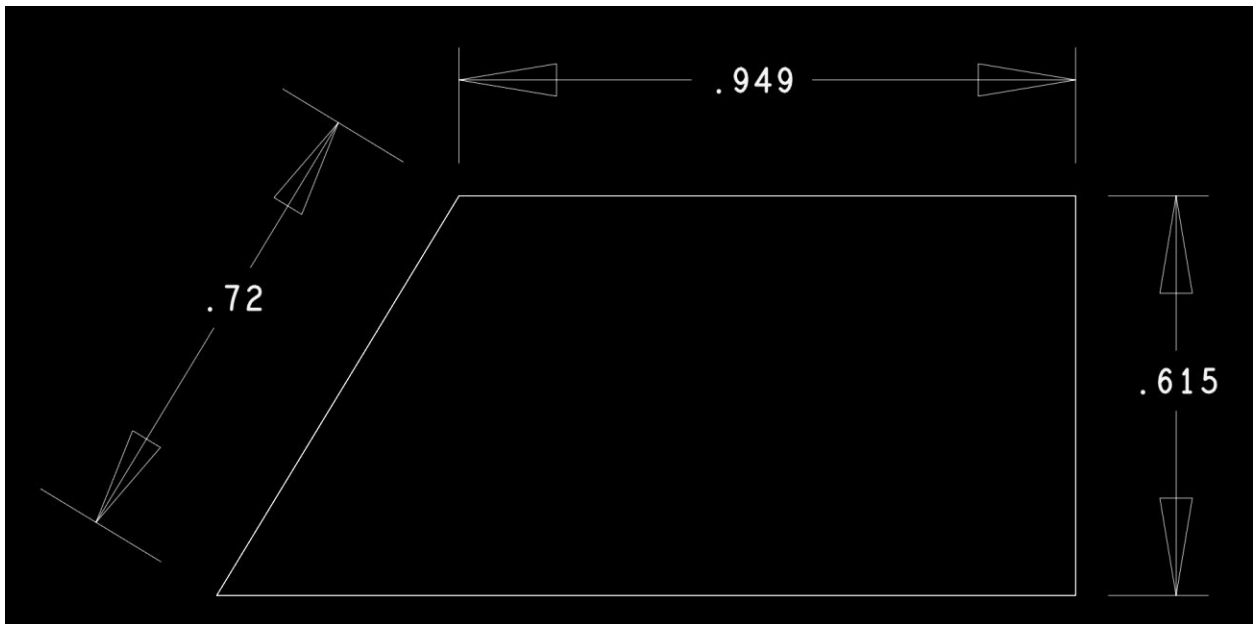

Figure 15: Linear dimensioning

Creating Linear Dimensions

1. Click **Manufacture > Dimension Environment**, and then right-click and select **Linear dimension**.
2. Position the cursor at the first point you want dimensioned (such as a corner of an element) and click.
3. Position the cursor at the second selection and click.
4. Select a position for the dimension.
5. To place the value in the center of the dimension line, enable **Center text in dimension lines.** The editor adds the dimension line and value.
6. Choose **Done** or **Next** from the pop-up menu to continue the command.

Datum Dimension

Datum dimensioning lets you specify dimensions relative to a user-defined reference point (datum). The dimensions are placed at the end of a single extension line. The extension line

can have multiple vertices, which lets you snake it through complex geometry to the outside of the design outline. This type of dimensioning is recommended for designs requiring a high density of dimensioning (see Figure 16).

The first dimension point you add to your layout is defined as x = 0 and y = 0. All other datum dimensions added are relative to the first point. You can override any dimension value by entering another value in the **Value** field in the **Options** tab.

When using the ANSI standard, you can rotate horizontal dimension text by 90 degrees by setting the **Align text with dimension line** parameter in the **Text** tab of the Dimensioning Parameters dialog.

Datum dimensioning enables you to control the element types (lines, arcs, rectangles, symbols, pins, and vias) to be dimensioned.
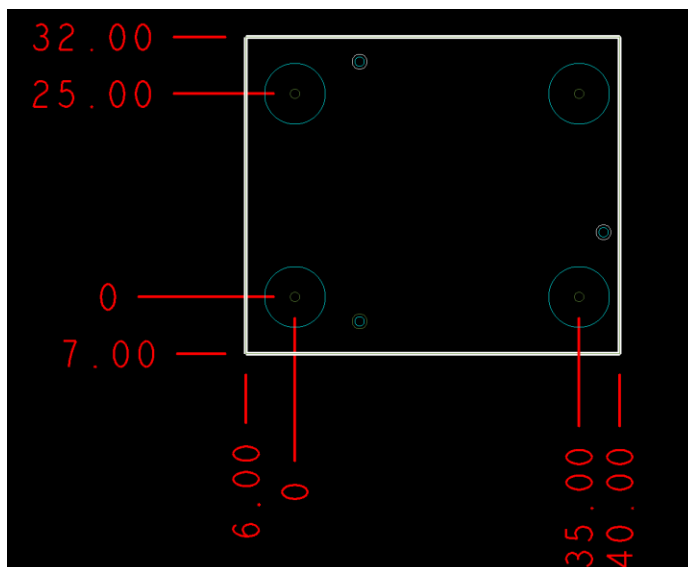

Figure 16: Datum dimensioning

You can choose datum dimensioning objects as a group or by using a window. If you choose the objects this way, dimensions are created for all line segment end points, arc and circle centers, symbol-instance origins, via and pin instances, and rectangle vertices according to the current settings in the **Find Filter**.

You can create windows of dimension instances, delete unnecessary instances, or create selective groups of dimensions. You can determine your dimensioning strategy to fit the requirements of each design. For ANSI, define borders for group/window sets by using rubber-banding border axis lines after making a group / window selection. For non-ANSI standards, the dimensions are aligned with reference datum targets.

Creating Datum Dimensions

1. Click **Manufacture > Dimension Environment**, and then right-click and select **Datum dimension**.

2. Click to select the first datum reference point.
    You can continue selecting extension line end points or choose **Dimension Value** (or Next) from the pop-up menu. The datum reference point appears (0).
    The **Dimension Axis** field in the **Options** tab shows the default. **Both**, along with axis choices **X** and **Y**.
    Always create a datum reference point using both axes.

3. Choose the second datum reference point.

4. Choose **Dimension Value** (or **Next**) from the pop-up menu to place the second datum reference point (0).

5. Once you have set a reference point for your datum dimensions, start selecting elements or locations and applying dimension values that show dimensions that are relative to the datum reference point that you created in steps 1 through 3.
    It may be easier to choose all datum dimensions one axis at a time. To do this, choose **Dimension Axis** in the **Options** tab. Set the axis to X, apply your dimensions, and set the axis to Y.

Angular Dimension

You can use this command to add angular dimensions between the line segments in your layout (see Figure 17). This command calculates the angle between chosen segments and creates the angular dimension lines and text required by the standard, specified in the **Dimensioning Parameters** dialog.

You can override the automatic dimension value by entering the appropriate value in the **Value** field on the **Options** tab before you choose the location for the text.
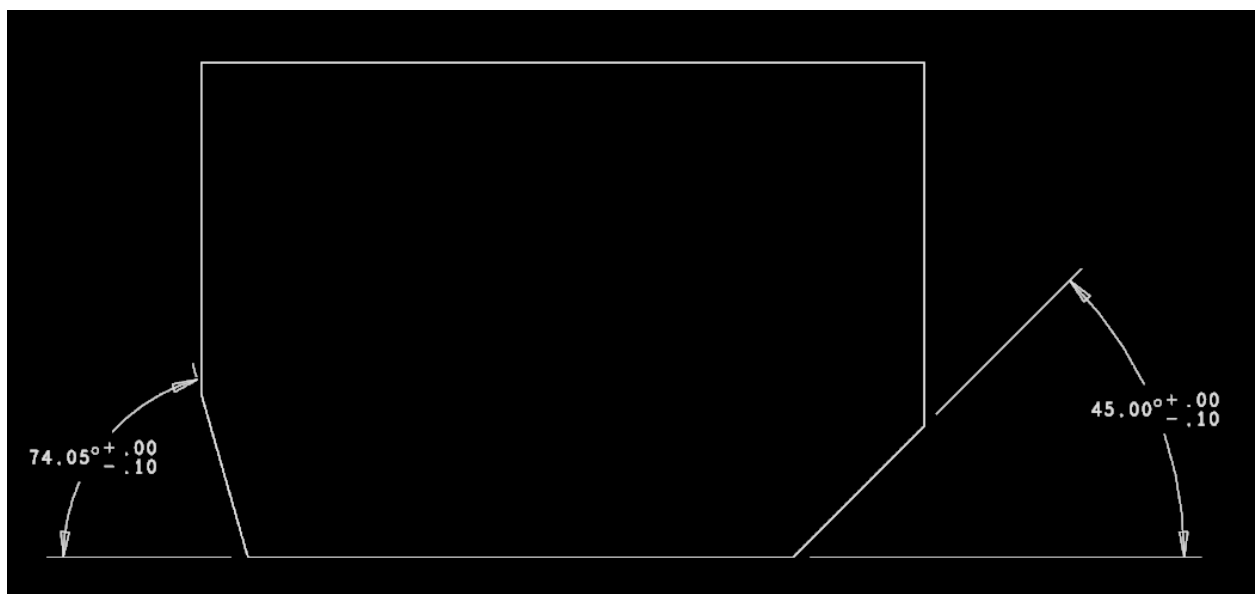
Figure 17: Angular dimensioning

Creating an Angular Dimension

1. Choose **Manufacture > Dimension Environment**, and then right-click to choose **Angular dimension**.
2. Click on one of the line segments to be dimensioned.
3. Select the second line segment to be dimensioned.
4. Select a position for the first extension line.
5. Select a position for the second extension line.
6. Select a location for the dimension. The editor adds the angular dimension lines and value.
7. Select **Done** or **Next** from the pop-up menu to continue with the command.

Leader-Oriented Dimensioning Modes

A leader is a note or dimension directed to the drawing feature for which it supplies information. Leaders are used for adding diametral and radial dimensions or balloons. Leaders also supply an alternative method for dimensioning 45-degree chamfers (see Figure 20).

The desired leader type can be specified in the RMB menu (see Figure 18).

Display characteristics for leaders (such as termination type and size) can be set in the Dimension lines section in the Lines tab (see Figure 19) of the Dimensioning Parameters dialog.
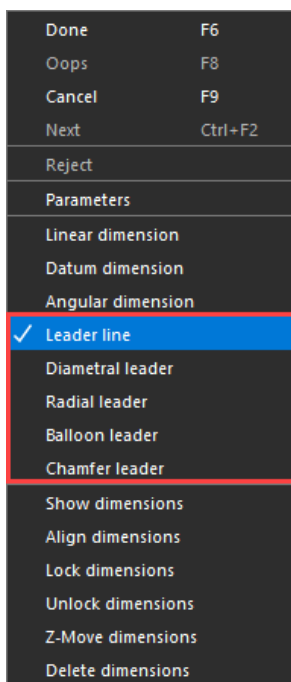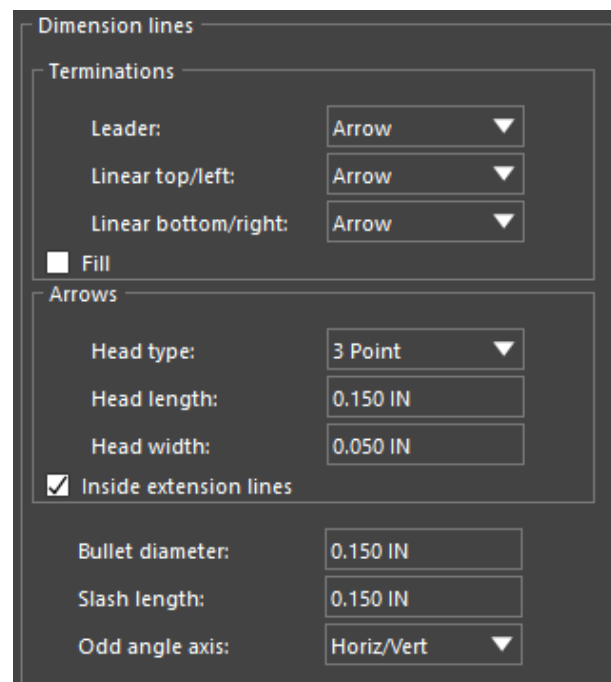

Figure 18: Leader type

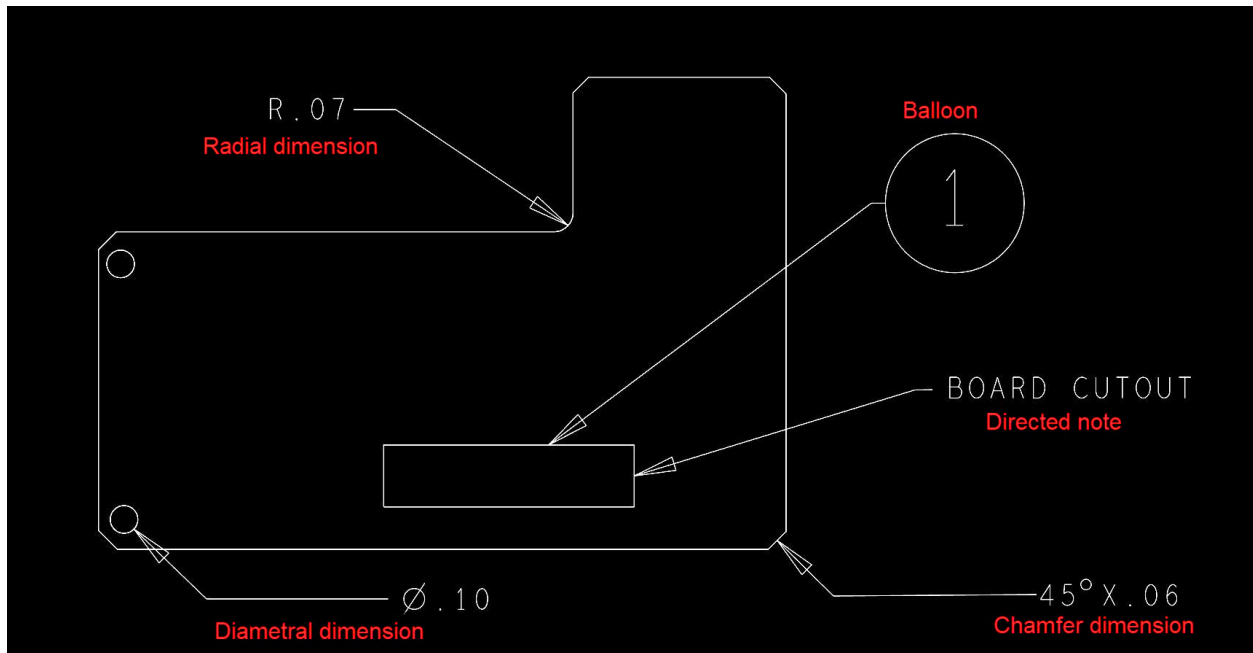Figure 19: Dimension lines settings

Figure 20: Leader-oriented dimensioning

Leader Line

A leader line dimension is added as a line (see Figure 21). To add a leader line, select **Leader line** from the RMB menu.

Figure 21: Leader line dimension
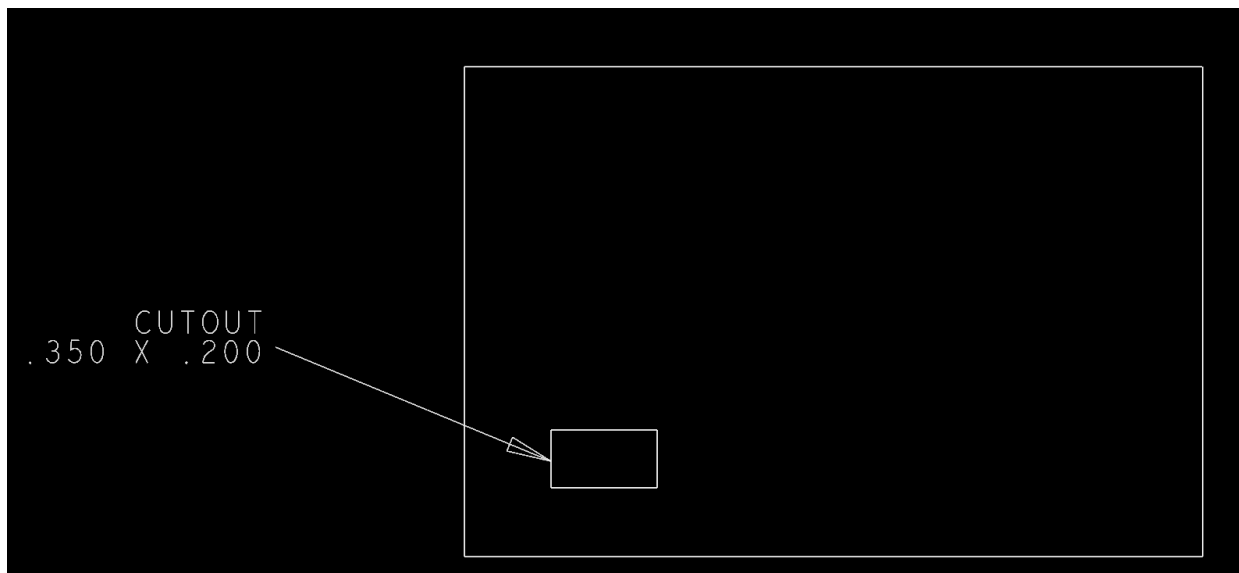
Diametral Leader

Diametral dimensioning identifies the diameter of a circle (see Figure 22). Select **Diametral leader** from the RMB menu to add a diametral leader.
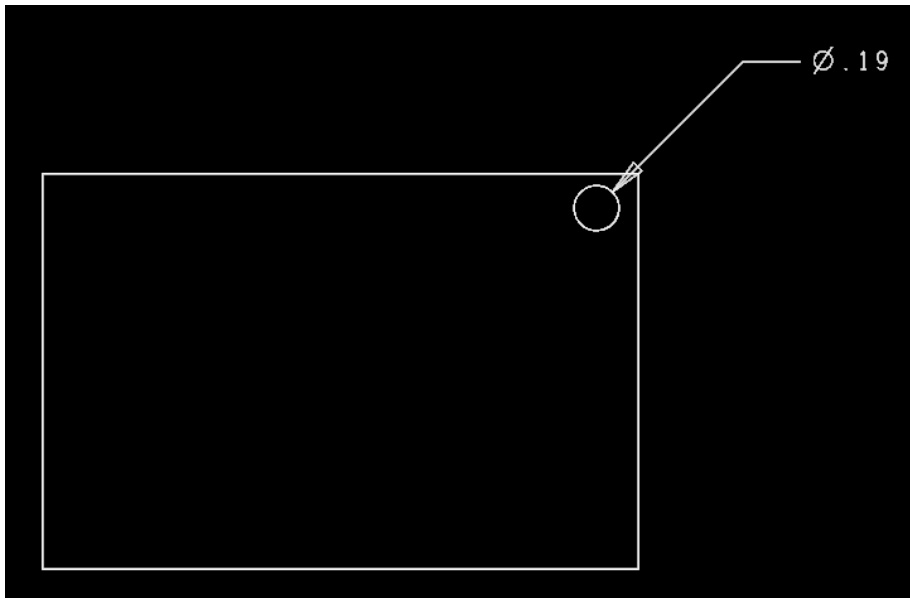

Figure 22: Diametral leader

Radial Leader

A radial leader identifies the diameter of a circle (see Figure 23). A radial leader can be added by selecting **Radial leader** from the RMB menu.
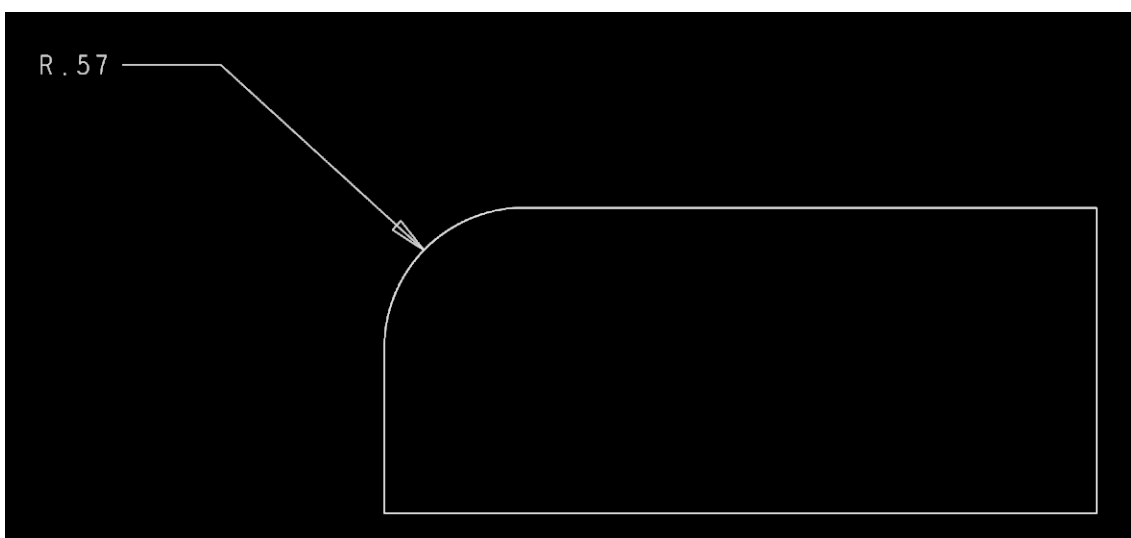

Figure 23: Radial leader

Balloon Leader

A balloon is a leader with a termination (arrow, bullet, or slash) on one end and a balloon (circle, square, triangle, or oblong) that encloses an alphanumeric character string on the other end (see Figure 24). Balloons typically point to an object, while the text enclosed in the balloon relates the object to another item. For example, it could be pointing to a component with the text inside the balloon relating the component to an item in a bill of materials. A balloon leader can be added by selecting **Balloon leader** from the RMB menu.
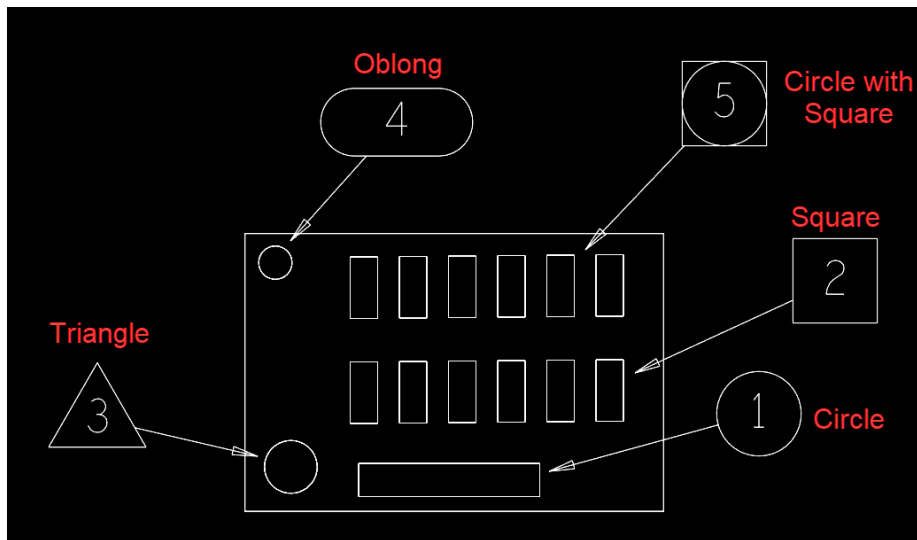

Figure 24: Balloon leader

Chamfer Leader

A chamfer leader provides an alternative and simpler way of dimensioning 45-degree chamfers than using a combination of linear and angular dimensioning (see Figure 25). A chamfer leader can be added by selecting **Chamfer leader** from the RMB menu.
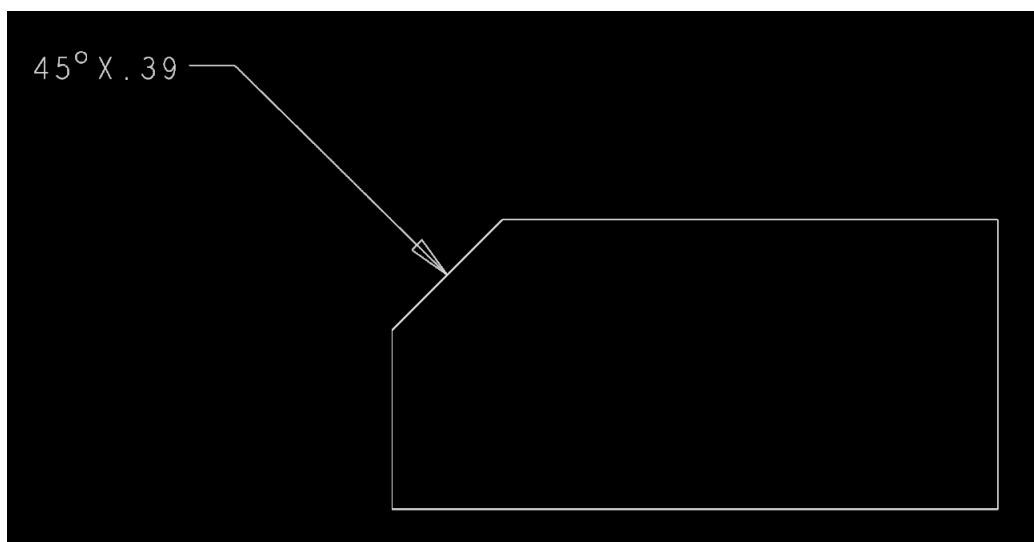

Figure 25: Chamfer leader

Show Dimensions

The **Show dimensions** mode shows dimension-related information of symbol objects (see Figure 26). You can select dimensions by a single pick, or by multiple picks with a window drag. Alternatively, you can choose **Select by Polygon** or **Temp Group** from the RMB menu.

Information given for a dimension symbol includes:

- Type of dimension
- Origin
- Class and subclass
- Object(s) with which the dimension is associated
- If the dimension is on a point in mid air
- If the dimension is locked
- If the dimension value is fixed

For a relative datum dimension, information is given on the reference datum object and is listed for each associated relative datum dimension object.
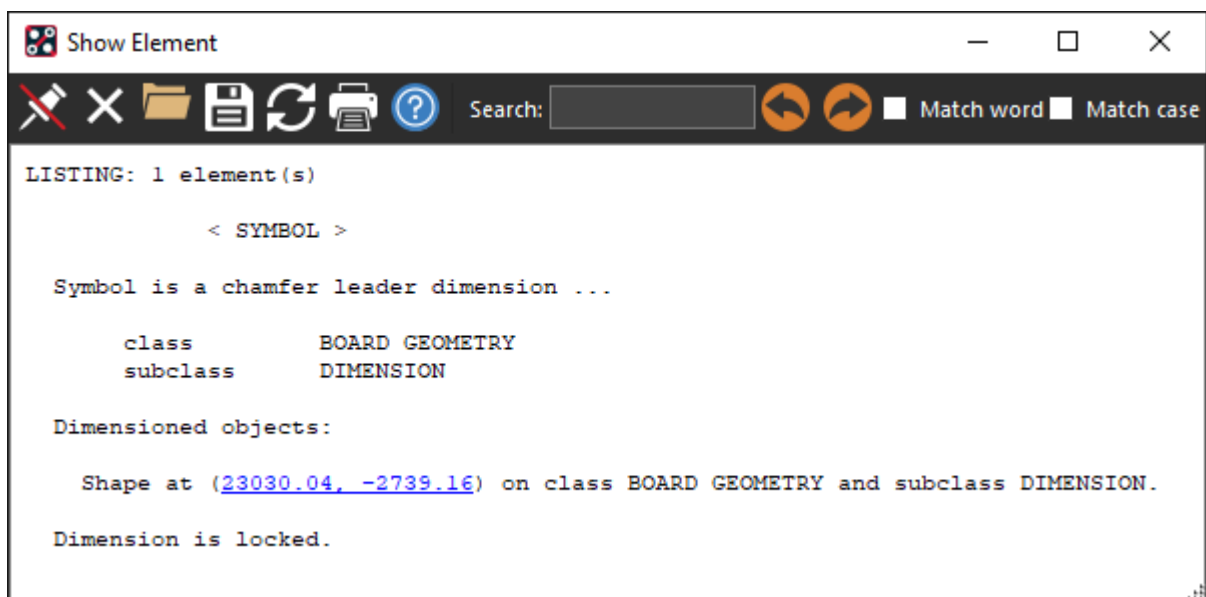
Figure 26: Show dimensions

Align Dimensions

The **Align dimensions** mode lets you align the dimensions with respect to the master dimension. The first dimension selected is a master dimension, and it remains fixed.

The dimensions selected subsequently are aligned to the master dimension. The dimension can be selected with a single pick or through window drag. Alternatively, you can select the dimension by choosing the **Select by Polygon** and **Temp Group** modes.

Aligning Linear and Datum Dimensions

Upon aligning, the text/endpoint locations of the dimensions are aligned. So, for linear dimensions in X direction, the Y locations of the dimension text are made the same as the master dimension and they are aligned in a row. Similarly, for linear dimensions in Y direction, the X locations are made the same as the master dimension to align them in a column. Linear dimensions in X cannot be aligned to a master linear Y dimension, and vice versa. The same rules for aligning linear dimensions in X and Y also similarly apply to datum dimensions in X and Y.

Aligning Leader Type of Dimensions

You can also align the leader dimensions. When you select the dimensions for aligning, a dialog appears allowing you to select the direction for alignment (see Figure 27).
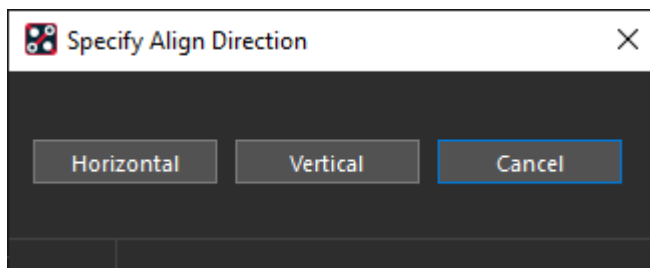

Figure 27: Align dimensions

If you select **Horizontal**, the text / endpoint locations of the leaders maintain their X locations and are aligned horizontally to the new Y locations of the master dimension, and vice versa.

You can lock the dimensions after aligning them using the **Lock dimensions** mode. These dimensions maintain their alignments with general editing of the objects they are associated with. You can align the locked dimensions also. In such cases, warning messages appear for the locked dimensions and they are re-locked at the new alignment locations.

Note

If any dimension selected for alignment is incompatible with the master dimension, a warning message is issued and the dimension is ignored.

Lock Dimensions

The **Lock dimensions** mode lets you fix the text location or leader end point on the board by locking the dimension. The dimension can be selected with a single pick or a window drag. You can also select the dimension by choosing the **Select by Polygon** and **Temp Group** modes. The leader types of dimensions are locked in both X and Y directions. For linear and datum dimensions, an X-axis dimension is locked in Y, and a Y-axis dimension is locked in X. If a dimension is already locked, a warning message is issued and the dimension is ignored.

Unlock Dimensions

The Unlock dimensions mode lets you unlock the dimensions. Selected dimensions are unlocked, and their current text end points float and move with the dimension.

Z-Move Dimensions

The **Z-Move dimensions** mode lets you move the selected dimensions to a new class and / or subclass. You can select the dimension with a single pick or by using a window drag. Alternatively, you can select the dimension by choosing the **Select by Polygon** and **Temp Group** selection modes.

Delete Dimensions

The **Delete dimensions** mode disassociates the selected dimensions from their objects and deletes them from the design. You can select the dimension with a single pick or by using a window drag. Alternatively, you can select the dimension by choosing the **Select by Polygon** and **Temp Group** selection modes.

Note

If a reference datum dimension is deleted, all its associated relative datum dimensions are automatically deleted.

Instance Parameters

The **Instance parameters** mode brings up the **Dimensioning Parameters** dialog and allows you to change only instance-specific parameters that are only applied to the selected dimension. The instance-specific settings initially shown reflect the last settings for the selected dimension. On closing the **Dimensioning Parameters** dialog, you can select the other dimension for changing its instance-specific parameters.

Move Text

The **Move text** mode moves the text of a dimension to a new location, and the dimension is recreated accordingly to accommodate the new text location. If you move a locked dimension, a waring message is issued saying that a new lock point is being set and you can move the locked dimension.

Mirror Text

The **Mirror text** mode mirrors the text of a dimension. You can also mirror the text of a locked dimension.

Change Text

The **Change text** mode lets you change the dimension text by entering the new value in the **Options** tab. If the current text value is fixed and you pick the dimension text to change, a warning message is issued. A fixed text value is reset to the actual computed value by entering **'0'** as the new text value in the **Value** field in the **Options** tab.

Edit Leaders

The **Edit leaders** functionality applies only to the leader types of dimensions and datum dimensions. This mode works with a single pick at a time. You can either edit an existing leader type of a dimension or create a new vertex at the pick point. When you select the pick point on the leader, a rubber-band graphic is attached to the cursor. You can select a new location for the vertex, and the dimension is recreated.

Delete Vertex

The **Delete vertex** mode lets you delete the current vertex from the dimension leader. This is a sub-mode of the **Edit leaders** mode and becomes temporarily selectable on the right-click when an existing or new vertex is attached to the cursor.

Summary

This application note covered the Dimension Environment capabilities that are available in Allegro PCB Editor, allowing you to have full control over the dimensioning processes which is necessary for the detail, and release-to-manufacturing of your design.