

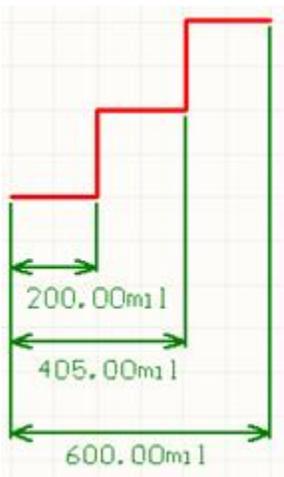
Baseline Dimension

Contents

- [Description](#)
- [Availability](#)
- [Placement](#)
- [Editing](#)
 - [Editing via an Associated Properties Dialog](#)
 - [Editing via the PCB Inspector Panel](#)
 - [Editing via the PCB List Panel](#)
 - [Graphical Editing](#)

Altium





Other Dimension Types

[Angular Dimension](#)

[Center Dimension](#)

[Datum Dimension](#)

[Leader Dimension](#)

[Linear Dimension](#)

[Linear Diameter Dimension](#)

[Radial Dimension](#)

[Radial Diameter Dimension](#)

[Standard Dimension](#)

Description

A baseline dimension is a group design object. It allows for the dimensioning of a linear distance of a collection of references, relative to a single base reference. The first point chosen is the 'base'. All subsequent points are relative to this first point. The dimension value in each case is therefore the distance between each reference point and the 'base', measured in the default units. The references may be objects (tracks, arcs, pads, vias, text, fills, polygons or components) or points in free space.

Availability

Baseline dimension objects are available for placement in the PCB Editor:

- select **Place » Dimension » Baseline** [shortcut: P, D, B] from the PCB Editor main menus
- click the  button on the Place Dimension drop-down of the Utilities toolbar.

Placement

After launching the command, the cursor will change to a crosshair and you will enter dimension placement mode. Placement is made by performing the following sequence of actions:

- position the cursor and click or press ENTER to anchor the dimension start point (this is the first reference point or 'base')
- move the cursor to the required end point and click or press ENTER to anchor the dimension end point (this is the second reference point)
- the text can now be initially positioned. Click or press ENTER when the text is in the desired position to effect placement
- move the cursor to subsequent reference points and click or press ENTER twice to effect placement (first click to anchor to a reference and second click after positioning the text)
- when all required references in the baseline dimension have been covered, right-click or press ESC to exit placement mode.

The baseline dimension object can be rotated during placement. Press the SPACEBAR to rotate the dimension anti-clockwise or SHIFT + SPACEBAR for clockwise rotation. Rotation is in accordance with the value for the Rotation Step, defined on the PCB Editor

- General page of the Preferences dialog (**Tools » Preferences**).

Editing

The properties of a baseline dimension object can be modified before, during and after placement. Editing itself falls into two categories

- graphical and non-graphical.

The following methods of non-graphical editing are available:

Editing via an Associated Properties Dialog

This method of editing uses the following dialog to modify the properties of a baseline dimension object:



Use the dialog's 'What's This Help' feature to obtain detailed information about each of the options available. Click on the question mark button at the top right of the dialog and then click over a field or option to pop-up information specific to that field or option. This dialog feature a units toggle control  in the top-left corner that will change the units of measurement currently used in the dialog between metric and imperial [shortcut: **CTRL + Q**]. The current unit of measurement is displayed in the dialog title area.

The Baseline Dimension dialog can be accessed prior to entering placement mode, from the PCB Editor - Defaults page of the Preferences dialog (**Tools » Preferences**). This allows you to change the default properties for the baseline dimension object, which will be applied when placing subsequent baseline dimensions.

During placement, the Baseline Dimension dialog can be accessed by pressing the TAB key.

After placement, the Baseline Dimension dialog can be accessed in the following ways:

- double-clicking on the placed baseline dimension object
- right-clicking the baseline dimension object and selecting Properties from the pop-up menu
- selecting the **Edit » Change** command, then clicking once over the placed baseline dimension object.

Editing via the PCB Inspector Panel

The PCB Inspector panel enables you to interrogate and edit the properties of one or more design objects in the active document. Used in conjunction with appropriate filtering, the panel can be used to make changes to multiple objects of the same kind, from one convenient location.



For more information on a specific panel, press F1 when the cursor is over the panel.

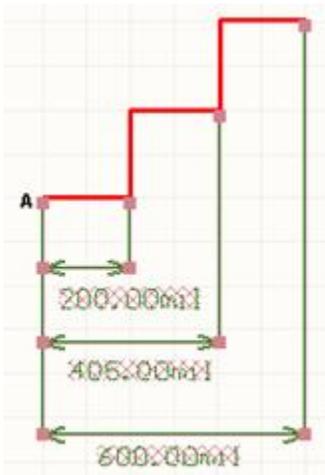
Editing via the PCB List Panel

The PCB List panel allows you to display design objects in tabular format, enabling you to quickly inspect and modify object attributes. When used in conjunction with the PCB Filter panel, it enables you to display just those objects falling under the scope of the active filter - allowing you to target and edit multiple design objects with greater accuracy and efficiency.



For more information on a specific panel, press F1 when the cursor is over a panel.

Graphical Editing



This method of editing allows you to select a placed baseline dimension object directly in the workspace and change properties such as the position of its text and its reference points, graphically.

When a baseline dimension object is selected, the following editing handles are available:

Click & drag the handles at arrows to adjust the dimension text position parallel to the extensions.

Click & drag A to move the base point of the dimension.

Click & drag subsequent handles to move each reference individually, with respect to the base.

All handles nearest to the object(s) being dimensioned allow for redefinable references

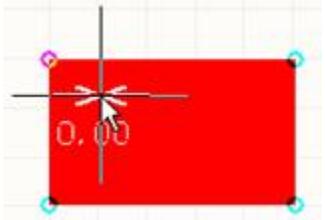
- once the dimension is detached from a reference object it becomes non-referenced and can be moved for attachment to a different reference point or object.

As you drag any of the editing handles, the dimension may be rotated.

If the baseline dimension object is totally non-referenced (i.e. it is not attached to any reference design objects) click anywhere on it

- away from editing handles

- and drag to reposition it. The dimension can be rotated or flipped while dragging. The latter is performed by pressing the X or Y keys to flip the dimension along the X-axis or Y-axis respectively.



Notes

When dimensioning an object, anchor points become available to you, highlighting where the dimension can be attached. Depending on the location of the cursor in relation to the object, one of the anchor points will highlight in a different color, specifying where the dimension will attach if you proceed to click or press ENTER.

You do not have to be exactly on the point for the anchor to highlight.

A baseline dimension object can be moved in the following ways:

- selecting both the dimension object and the object that is being dimensioned. The whole can be dragged to a new location as required
- selecting an object that is being dimensioned only. The dimension text will follow the object in its alignment plane only. The dimension extensions will expand/contract to keep the relationship between dimension and object being dimensioned
- selecting the dimension object only. It is important to note that the dimension cannot be moved on its own if it is referenced by a design object. To move the dimension only, it must first be detached from the object it is dimensioning.

If you attempt to graphically modify a baseline dimension object that has its Locked property enabled, a dialog will appear asking for confirmation to proceed with the edit.

Dimension text is rendered using either a Stroke or TrueType font. Three Stroke-based fonts are available - **Default** , **Sans Serif** and **Serif** . The **Default** style is a simple vector font which supports pen plotting and vector photoplotting. The **Sans Serif** and **Serif** fonts are more complex and will slow down vector output generation, such as Gerber. The Stroke-based fonts are built into the software and cannot be changed. All three fonts have the full IBM extended ASCII character set that supports English and other European languages

The **Default** Stroke font is used by default. Change to another Stroke-based font or enable the TrueType option in the Designator or Comment dialog as required. In the latter case, the **Select TrueType Font** options will become available.

Select the particular TrueType font you wish to use from the Font Name list (populated with TrueType and OpenType (a superset of TrueType) fonts found in the *Windows\Fonts* folder). Note that the list will only include entries for detected (and uniquely named) root fonts. For example, *Arial* and *Arial Black* will be listed but *Arial Bold* , *Arial Bold Italic* , etc will not. Use the Bold and Italic options to add emphasis to the text. The feature also offers full Unicode support.

Use the available save/load options on the PCB Editor - TrueType Fonts page of the Preferences dialog to enable embedding of TrueType fonts when saving a design, and for nominating a substitution TrueType font for files using TrueType fonts that are not available installed locally.

The dimension value automatically updates as you move the start or end points. Likewise, if the position of the object that either reference point of the dimension is anchored to is changed, the dimension will update and expand/contract to reflect this.

When the reference or references to which a dimension object is attached are deleted, a dialog will appear, asking whether the dimension should also be deleted. If the dimension is not deleted, it remains on the PCB sheet, but non-referenced.

Baseline dimensions are group objects consisting of text and track segments. They can be converted to their set of primitive objects by choosing **Tools » Convert » Explode Dimension to Free Primitives** from the main menus. Once exploded, a dimension object can no longer be manipulated as a group object.

Any changes made to object properties during placement will cause the default properties for the object to be updated, unless the Permanent option

- on the PCB Editor

- Defaults page of the Preferences dialog

- is enabled. When this option is enabled, changes made will affect only the object being placed and subsequent objects placed during the same placement session.