

Coordinate

Modified by Jason Howie on Sep 2, 2014

Parent page: [Objects](#)



A placed Coordinate.

Summary

A coordinate is a group design object. It is used to mark the X (horizontal) and Y (vertical) distance of a point in the design workspace relative to the current origin. Coordinates can be placed on any layer.

Availability

Coordinates are only available for placement in the PCB Editor:

- Click **Home** | **Place** |  **Coordinate**.

Placement

After launching the command, the cursor will change to a cross-hair and you will enter coordinate placement mode. A coordinate will appear "floating" on the cursor:

1. Position the cursor and click or press **Enter** to place a coordinate.
2. Continue placing further coordinates, or right-click or press **Esc** to exit placement mode.

Additional actions that can be performed during placement are:

- Press the **L** key to flip the coordinate to the other side of the board.
- Press the + and - keys (on the numeric keypad) to cycle forward and backward through all visible layers in the design respectively - to change placement layer quickly.
- Press the **Tab** key to access an associated properties dialog, from where properties for the coordinate can be changed on-the-fly.

Graphical Editing

This method of editing allows you to select a placed coordinate object directly in the workspace and

change its location graphically. The size and shape of a coordinate object cannot be changed graphically. As such, editing handles are not available when the coordinate object is selected:



A selected Coordinate.

- Click anywhere on the coordinate and drag to reposition it. The position values are automatically updated as the coordinate is moved.

An object that has its **Locked** property enabled cannot be selected or graphically edited. Double click on the locked object directly and disable the **Locked** property, to graphically edit the object.

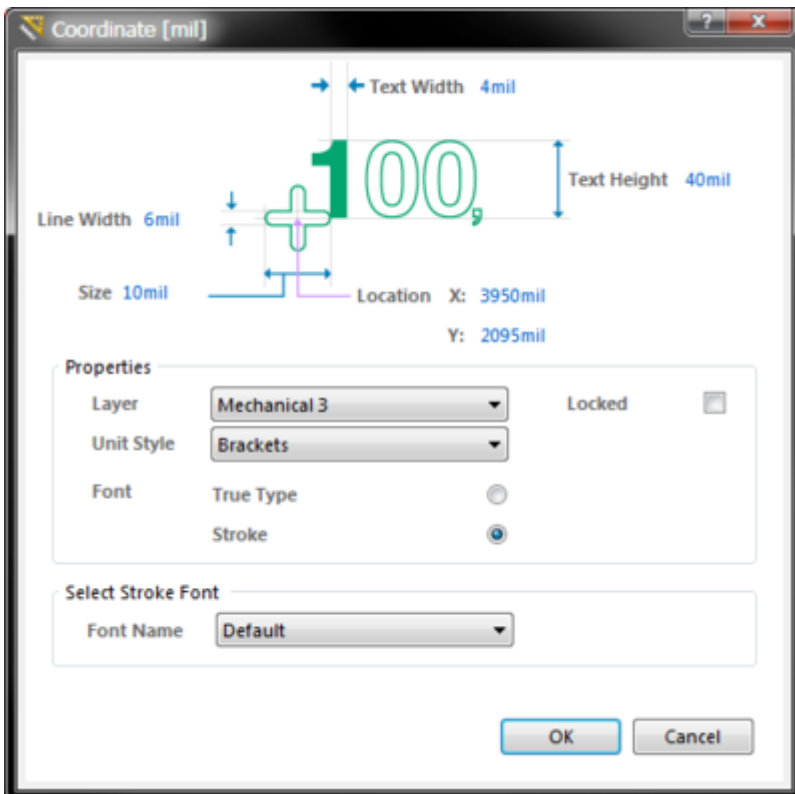
Non-Graphical Editing

The following methods of non-graphical editing are available:

Editing via an Associated Properties Dialog

Dialog page: [Coordinate](#)

This method of editing uses the [Coordinate dialog](#) to modify the properties of a coordinate object.



The *Coordinate* dialog.

The *Coordinate* dialog can be accessed prior to entering placement mode, from the **PCB Editor - Defaults** page of the *Preferences* dialog. This allows the default properties for the coordinate object to be changed, which will be applied when placing subsequent coordinates.

During placement, the dialog can be accessed by pressing the **Tab** key.

After placement, the dialog can be accessed in one of the following ways:

- Double-clicking on the placed coordinate object.
- Placing the cursor over the coordinate object, right-clicking and choosing **Properties** from the context menu.

Editing via an Inspector Panel

An *Inspector* panel enables the designer 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.

Notes

1. The current origin can be set to be any point in the PCB workspace by selecting **Home | Grids and Units | Origin » Set**, and clicking on the location for the new origin. To set the current origin back to the absolute origin ((0, 0) in the lower-left corner of the workspace), select **Home | Grids and Units | Origin » Reset**.

Source URL: [http://documentation.circuitmaker.com/display/CMAK/PCB_Obj-Coordinate\(\(Coordinate\)\)_CM](http://documentation.circuitmaker.com/display/CMAK/PCB_Obj-Coordinate((Coordinate))_CM)