Cartesian Grid Editor
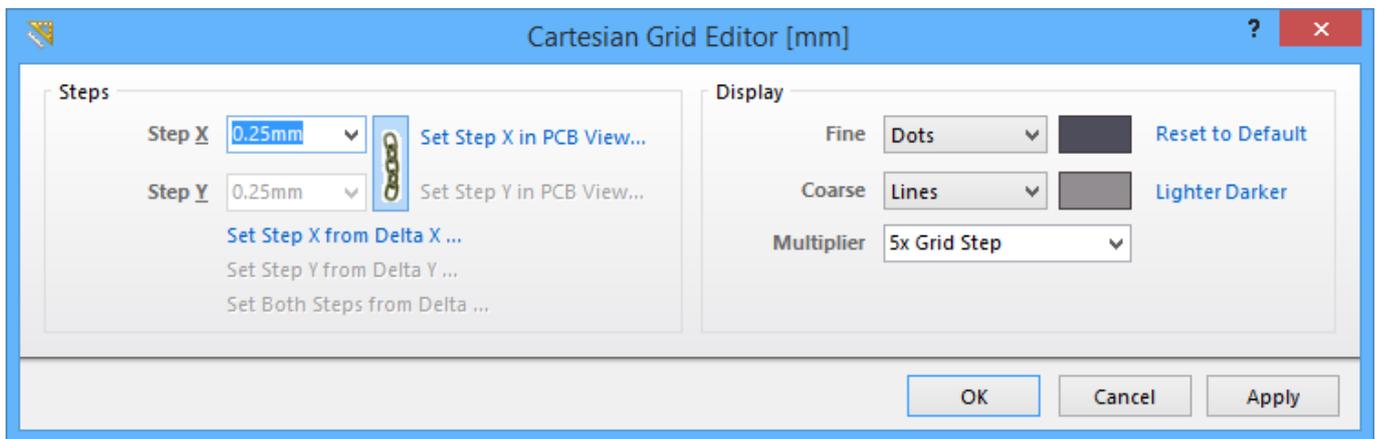
Modified by Annika Krilov on Jun 23, 2015

Other Related Resources
[Grids and Units](#)

Parent page: [PCB Dialogs](#)

The *Cartesian Grid Editor* dialog.

Summary

This dialog allows the designer to view and modify properties for the default snap grid defined for the board. The default snap grid - named `Global Board Snap Grid` - is used for object placement and movement in any area of the board. This Cartesian-type grid can be modified with respect to its step size, and display.

Access

The dialog can be accessed from both the PCB Editor, and PCB Library Editor:

- *PCB Editor* - click **Home | Grids and Units |** 🔧 Properties, from the main menus. Alternatively, right-click in the workspace and use the **Snap Grid » Grid Propereties** command from the context menu.
- *PCB Library Editor* - click **Home | Grids and Units | Grids » Properties**, from the main menus. Alternatively, right-click in the workspace and use the **Snap Grid » Grid Propereties** command from the context menu.

✅ The keyboard shortcut - **Ctrl+G** - can also be used in either editor to quickly access the dialog.

Options/Controls

Display

- **Fine** - use the associated drop-down to define the markers used for the fine-level display of the grid in the workspace, either `Lines` or `Dots`. The step size used for the markers is that defined in the **Steps** region. Click on the associated color swatch to access the standard *Choose Color* dialog, from where you can specify the color to be used for the fine-level display grid in the workspace. You can always reset the color back to its default using the **Reset to Default** link.
- **Coarse** - use the associated drop-down to define the markers used for the coarse-level display of the grid in the workspace. Again, choose from either `Lines` or `Dots`. The coarse-level display grid is simply the fine-level display grid with an increased step size, in accordance with the entry selected in the **Multiplier** field. If you don't want to use the coarse-level display grid, simply choose the option `Do Not Draw`.

  Click on the associated color swatch to access the standard *Choose Color* dialog, from where you can specify the color to be used for the coarse-level display grid in the workspace. You are free to choose a completely different color to that used for the fine-level display grid. Alternatively, you can quickly employ a lighter or darker shade of the color currently used for the fine-level display grid, using the available **Lighter** or **Darker** links respectively. Again, you can reset the color back to its default using the **Reset to Default** link.

- **Multiplier** - use this field to specify the required multiple of the grid's step size, either 2x `Grid Step`, 5x Grid Step, or `10x Grid Step`.

> The default display colors – assigned to **Fine** and **Coarse** display grids when the **Reset to Default** link is clicked – are defined in the **System Colors** region, on the **Board Layers And Colors** tab of the *View Configurations* dialog (**View | View |** Switch to 3D **» View Configuration**). Specifically, the default fine-level display grid color is specified using the **Default Grid Color - Small** option, and the default coarse-level display grid color is specified using the **Default Grid Color - Large** option.

Steps

- **Step X** - use this field to define the distance between grid lines in the X plane. Type the required step size directly, or select from a range of common sizes available in the associated drop-down list.
- **Step Y** - use this field to define the distance between grid lines in the Y plane. Type the required step size directly, or select from a range of common sizes available in the associated drop-down list.

> By default, the two fields are linked, as indicated by the continuous chain depicted on the button to the right of the fields - . In this state, whatever you specify for the **Step X** field will be copied and used for the **Step Y** field. To break this link and enter step sizes individually, simply click this button. The button will now depict a broken chain, and the **Step Y** field will become available for editing in its own right.

The following controls are also available that allow you to define the X and/or Y step sizes directly from within the PCB workspace. In each case, you will be taken to the workspace to specify two

'calculating' locations, and the resulting step size will be calculated accordingly.

- **Set Step X in PCB View** - the resulting size is taken as the hypotenuse of the triangle formed by the chosen points in the workspace.
- **Set Step Y in PCB View** - the resulting size is taken as the hypotenuse of the triangle formed by the chosen points in the workspace.
- **Set Step X from Delta X** - the resulting size is taken using just the difference in the X coordinate.
- **Set Step Y from Delta Y** - the resulting size is taken using just the difference in the Y coordinate.
- **Set Both Steps from Delta** - the resulting sizes are taken using just the differences in the X and Y coordinates respectively.

> Note that when **Step Y** is following **Step X** (chain linked), only the **Set Step X in PCB View** and **Set Step X from Delta X** controls will be available.