

Snap Guide Manager

Modified by Jason Howie on May 8, 2015

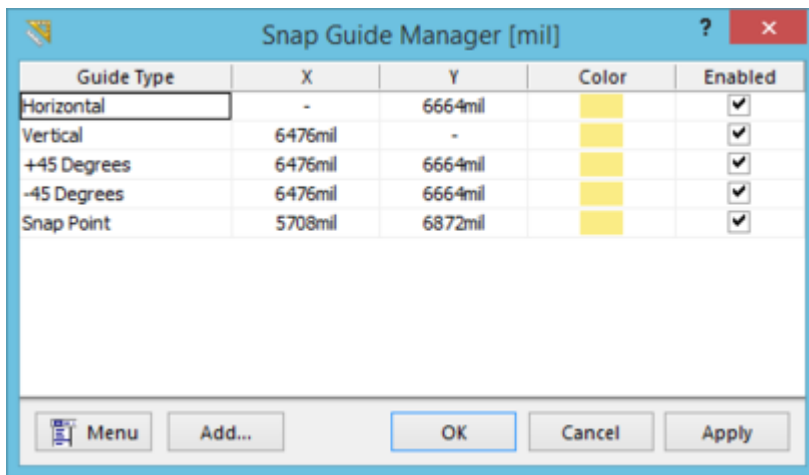
Other Related Resources

[Cartesian Grid Editor \(Dialog\)](#)

[Grids and Units](#)

[Board Options \(Dialog\)](#)

Parent page: [PCB Dialogs](#)



The *Snap Guide Manager* dialog.

Summary

This dialog provides the designer with a centralized location from which to define and manage all *Snap Guides* for the active PCB document – both *Linear Snap Guides* and *Point Snap Guides* (or manual *Snap Points*, if you will).

Snap guides are special objects that are manually placed specifically for the purpose of driving the cursor-snap on a certain axis or point – assisting in object/component placement. They can also serve as a visual indicator for general layout or alignment purposes. During an interactive process such as placing or moving, the cursor will snap to a placed guide, at the point where that guide intersects the default snap grid. Using a guide, objects can quickly be aligned simply by dragging them until they 'snap' against the guideline.



The cursor will only snap to placed linear and point snap guides, provided the **Snap To Linear Guides** and **Snap To Point Guides** options are enabled respectively, in the **Snap Options** region of the *Board Options* dialog.

Access

The dialog can be accessed from both the PCB Editor, and PCB Library Editor, by clicking the **Guides** button, at the bottom of the *Board Options* dialog.

Options/Controls

- **Defined Guides List** - this, the main area of the dialog, presents, and allows the management of, all currently defined snap guides. The following information is presented for each guide:
 - **Guide Type** - either Vertical, Horizontal, +45 Degrees, -45 Degrees, or Snap Point.
 - **X** - the x-coordinate (where applicable) specified in the workspace that the guide is to pass through, or point is to be located at.
 - **Y** - the y-coordinate (where applicable) specified in the workspace that the guide is to pass through, or point is to be located at.

 A coordinate can be defined by clicking on the relevant field, and entering the value. Alternatively, and where applicable/available, click the  button to the right of the field, then specify the required point graphically, and directly, in the PCB workspace.

 - **Color** - the color used for the guide/point. Double-click on the color swatch to access the standard *Choose Color* dialog, from where you can assign any color as required. To apply the same color to all existing guides in 'one shot', use the **Set Color for All** command, available from the dialog's main and right-click menus.
 - **Enabled** - whether the guide is visible in the workspace (checked) or not (unchecked).
- **Add** - click this button to add a new snap guide or snap point. Choose the corresponding command for the required guide type from the associated menu, an entry for the new guide/point will be added to the list. The following guide types are available:
 - **Add Horizontal Guide** - use this command to place a horizontal guideline at the desired Y-coordinate location in the workspace.
 - **Add Vertical Guide** - use this command to place a vertical guideline at the desired X-coordinate location in the workspace.
 - **Add +45 Guide** - use this command to place a 45 degree ($y=x$) guideline that passes through the desired X,Y coordinate location in the workspace.
 - **Add -45 Guide** - use this command to place a -45 degree ($y=-x$) guideline that passes through the desired X,Y coordinate location in the workspace.
 - **Add Snap Point** - use this command to place a point snap guide. This is simply a hotspot that you manually mark within the confines of the default snap grid. During an interactive process, such as placing or moving an object, that objects' hotspot will 'snap' to a point snap guide, when it passes into close proximity with it.
- **Menu** - click this button to access the following commands, all of which can also be found on the defined guides list's right-click menu:
 - **Set Color for All** - use this command to quickly set all guides and points to use the same chosen color. The standard *Choose Color* dialog will appear, with which to do so.
 - **All Enabled** - use this command to quickly enable the display of all defined guides and points (**Enabled** option checked for all).
 - **All Disabled** - use this command to quickly disable the display of all defined guides and points (**Enabled** option unchecked for all).

- **Toggle Selected** - use this command to quickly toggle the display of all selected guides and points (**Enabled** option toggles for selected entries in the list only).
- **Delete Selected** - use this command to quickly delete all guides and points currently selected in the list.
- **Delete All** - use this command to quickly delete all guides and points in the list.
- **Duplicate Selected** - use this command to quickly duplicate the currently selected guide or point in the list. The duplicated entry will be added to the bottom of the list. Note that only a single existing guide or point may be duplicated by this action, but there is no limit to the number of duplicate actions that can be performed.
- **Import** - use this command to import a PCBGuide file with previously exported guides and points. The *Load Guides* dialog will appear. Use this to determine the PCBGuide file (*.PCBGuide) to be imported. The imported guides and points will be appended to those currently presented in the list.
- **Export Selected** - use this command to export the guides and points that are currently selected in the list. The *Save PCBGuides* dialog will appear. Use this to determine where, and by what name, the exported PCBGuide file (*.PCBGuide) is to be saved.
- **Export All** - use this command to export the currently defined list of guides and points. The *Save PCBGuides* dialog will appear. Use this to determine where, and by what name, the exported PCBGuide file (*.PCBGuide) is to be saved.
- **More Information** - use this command to access this page of the documentation.

Source URL:

[http://documentation.circuitmaker.com/display/CMAK/PCB_Dlg-GuideManagerForm\(\(Snap+Guide+Manager\)\)_CM](http://documentation.circuitmaker.com/display/CMAK/PCB_Dlg-GuideManagerForm((Snap+Guide+Manager))_CM)