Title: Color and Visibility

Product: OrCAD / Allegro PCB Editor

Summary: This application notes describes the different color and visibility settings. As well the color views settings are explained, which helps to change the different view's quickly during daily work.
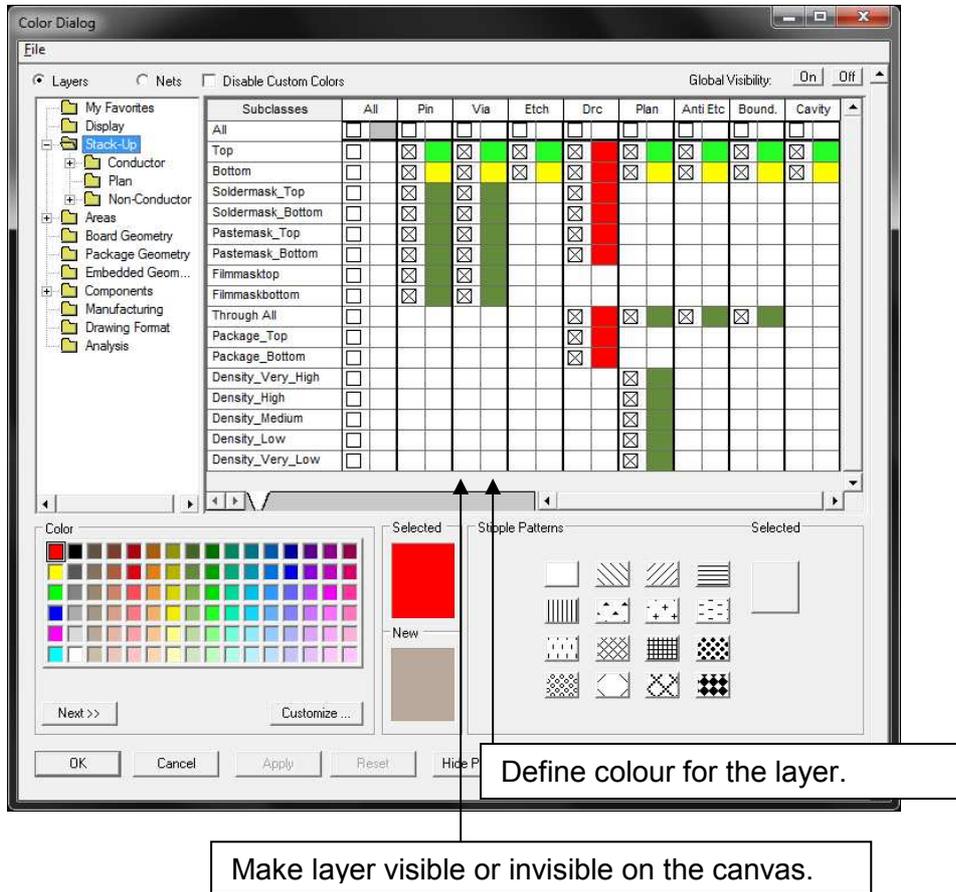
Author/Date: Marco Waller / 25.09.2012

Table of Contents

1 Define Colors and Patterns

With the Display→ Color/Visibility you can launch the color GUI. The same GUI can be

launched as well with following Icon:

Define colour for the layer.

Make layer visible or invisible on the canvas.

In this window you can define the color and visibility for each layer separately. Additionally to the colors, you can also define some patterns for each layer. With the patterns you have more flexibility when coloring layers.

In following example, the Bottom /Etch Layer have associated a pattern.
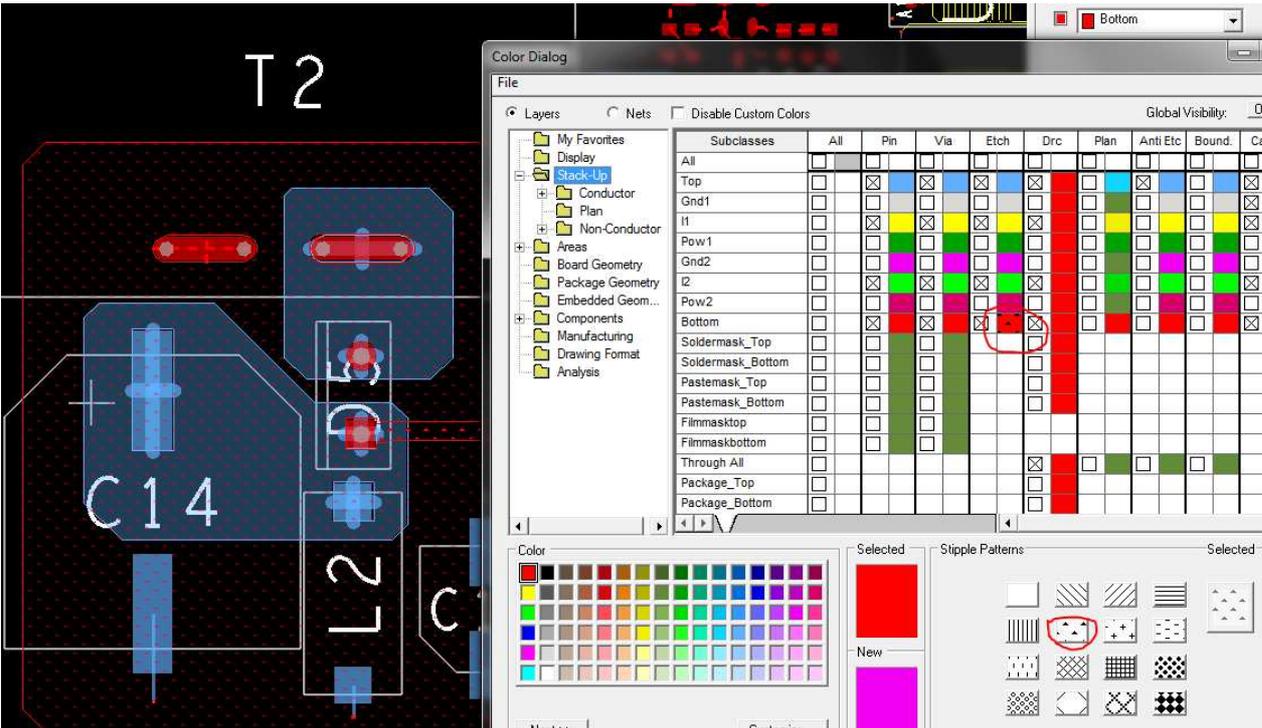

1.1 Visibility window

The visibility window allows you to easily activate or deactivate a layer in the canvas. When selecting one of the little rectangles, the visibility can be modified. The black square means the layer is invisible, the colored rectangle means that this layer is visible. The color matches the settings which were made in the color window.

Layer is invisible on the canvas.

Click on the rectangle to make this layer visible!
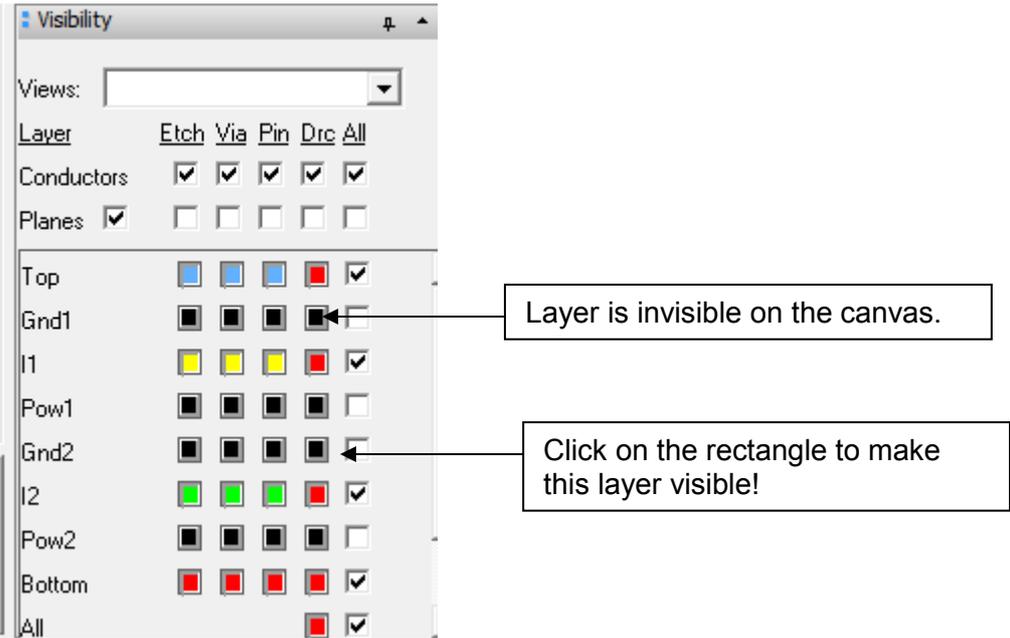
2 Color Views

The Color views give the opportunity to change the display in the canvas rapidly. First of all, the color views have to be created with the command View → Color View Save.

A new window appears. In the "save view:"- field, a name has to be specified for the view. Different options can be set which are explained below. With the save command, a color view file (example: top.color) is created in the current working directory.
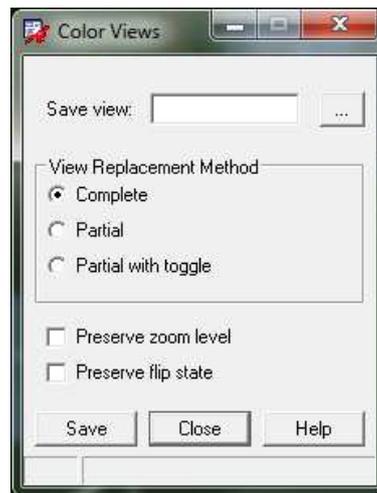

| *Complete*	Saves the current layer visibility settings to a color view file. When you load the file later, it completely replaces the design's visibility settings, analogous to how the film option to color views works.
Partial	Allows a color view file to be created that stores only *changes* to visibility settings. A partial color view does not replace all of a design's visibility settings when loaded. It only replaces the items you changed when you created the color view file. For example, if you changed the color for all DRCs to visible, when you loaded that color view file into a design, only the DRCs would be affected, all changing to visible. All other visibility settings would remain unchanged.
Partial with toggle	Functions the same as the *Partial* view replacement method because the color view file only stores changes. However, settings that you change toggle when you load the color view file. Toggle means that if the visibility for a layer is on in a design, when you load the color view file, it is turned off. If off, it is turned on.
Preserve Zoom Level	Includes the zoom points of the current design with the color view.

Create all you're desired views manually. Afterwards you can store all the *.color file on a network drive (ex.: <Install_Directory>/Cadence/settings/views). In the next step you can define the path in the PCB Editor, where the color-views are saved. Under setup→ User preferences→Paths → Config → Viewpath add the path to the list.


Search for path.
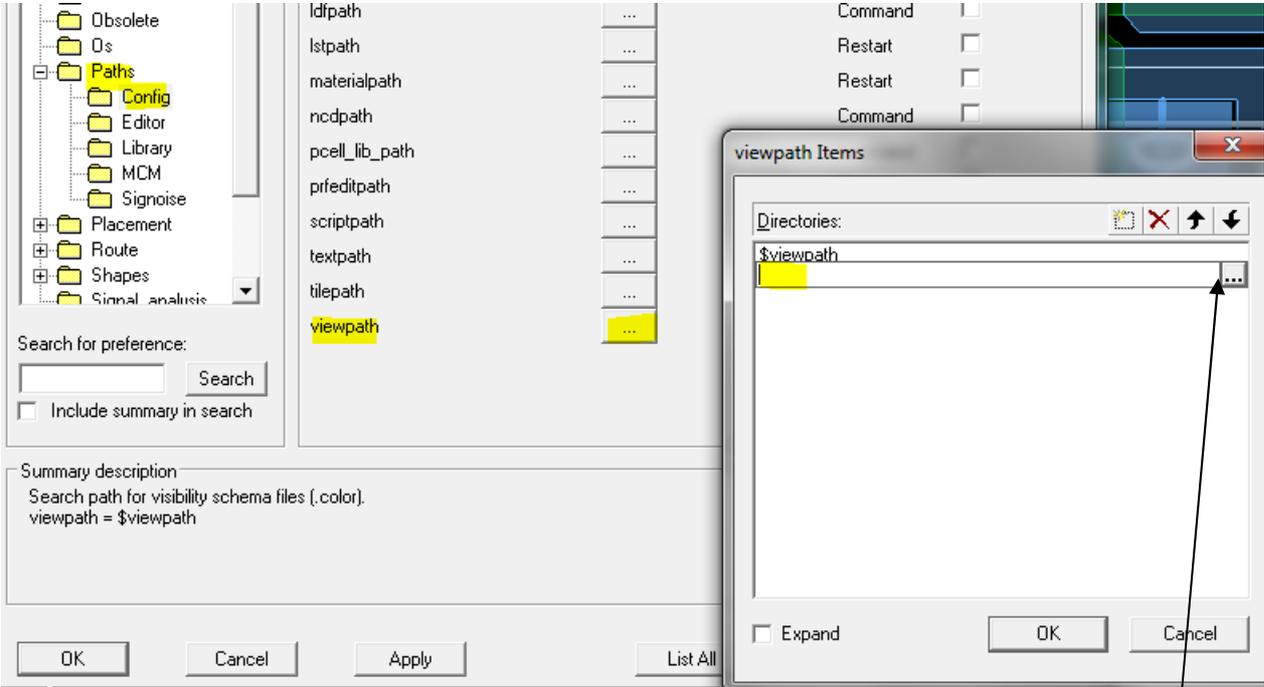
After restarting the PCB Editor, all the different color views can be selected in the visibility window under views.
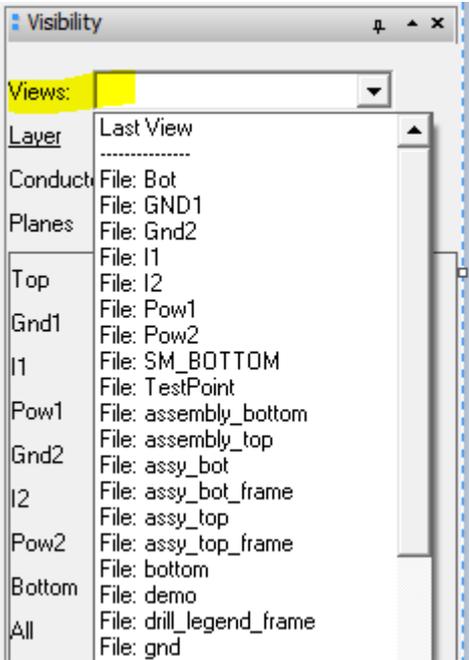
2.1 Color view: Partial with toggle

This chapter describes the "Partial with toggle" view more detailed. Imagine you've defined test points on your board. Then you desire that the TP layer can be toggled with a color view function. Here's the description how to create such a view:

1. Open the view → Color view window
2. In the Save view filed open the directory where you stored all the other *.color files
3. Enter a name in the filename label, for example: TP_Toggle
4. Chose the Option "Partial with toggle" option
5. Open the color/visibility dialog
6. Enable the Probe_Bottom and Probe_Top class in the "Manufacturing" Folder.
7. Close the color/visibility dialog
8. Select Save in the Color Views window and close the window afterwards
9. In the visibility window you can now select the TP_Toggle view, which makes the TP visible or invisible on the active canvas
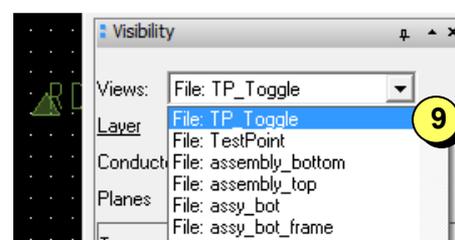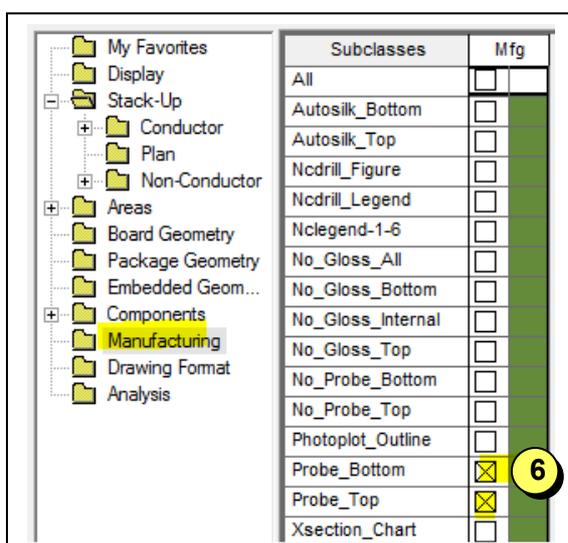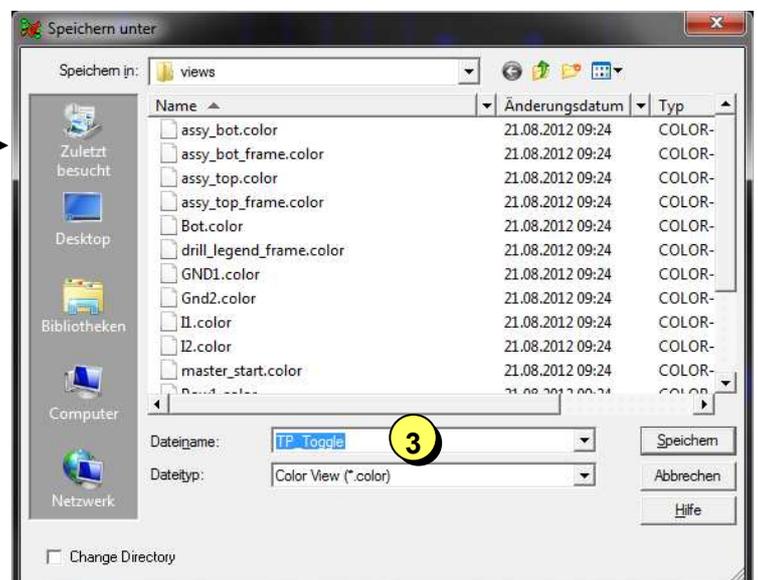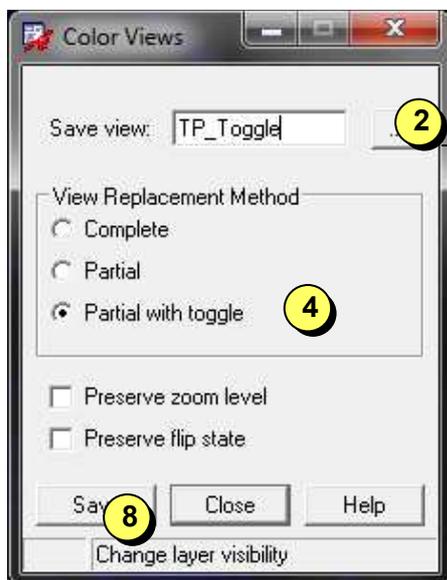
3 Export / Import Color settings

It is also possible to export or import the color settings. So you can easily reuse your predefined color settings. In the Export→ Parameters dialog you can chose, which settings you want to export.
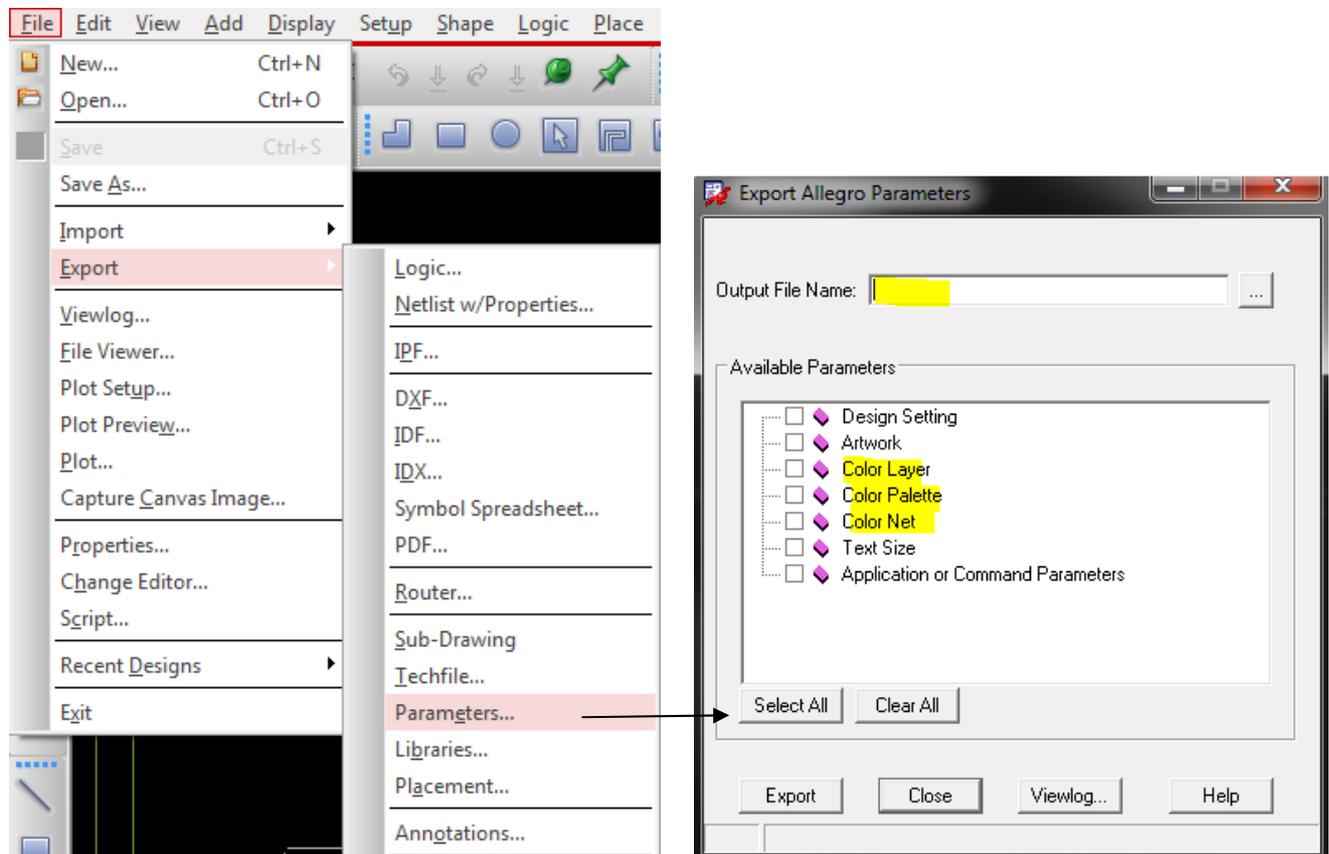
Color Layer	Priority in which layers are drawn
Color Palette	Color parameters and color table
Color Net	Net custom color and states. When a file containing net color data is imported into any design, only the nets that exist in that design are read; the rest are ignored. Net color assignments are not overwritten, but rather incremented. To completely replace net color assignments, click *Clear All Nets* in the *Nets* section of the Color dialog box before importing a file containing net color data.

4 Additional Information

For more information please contact FlowCAD:

Switzerland	Germany	Poland
+41 (0)56 485 91 91	+49(89) 4563-7770	+48 (58) 732 74 77
support@FlowCAD.ch	support@FlowCAD.de	support@FlowCAD.pl