# Introduction

PCB Editor has the ability for users to create and use Colour View Files. These are default views that users need on a daily basis. Without the colour view files users have to use the Colour Visibility GUI or the quick view tab to turn layers on an off. Creating and storing a colour view file enables users to create specific view files, add them to a default network location and then use them quickly.

How to Create a Color View File.

Use Display > Color/Visibility (Allegro) or Setup > Colors (OrCAD) to launch the colour GUI. Setup the colours as per your requirement.
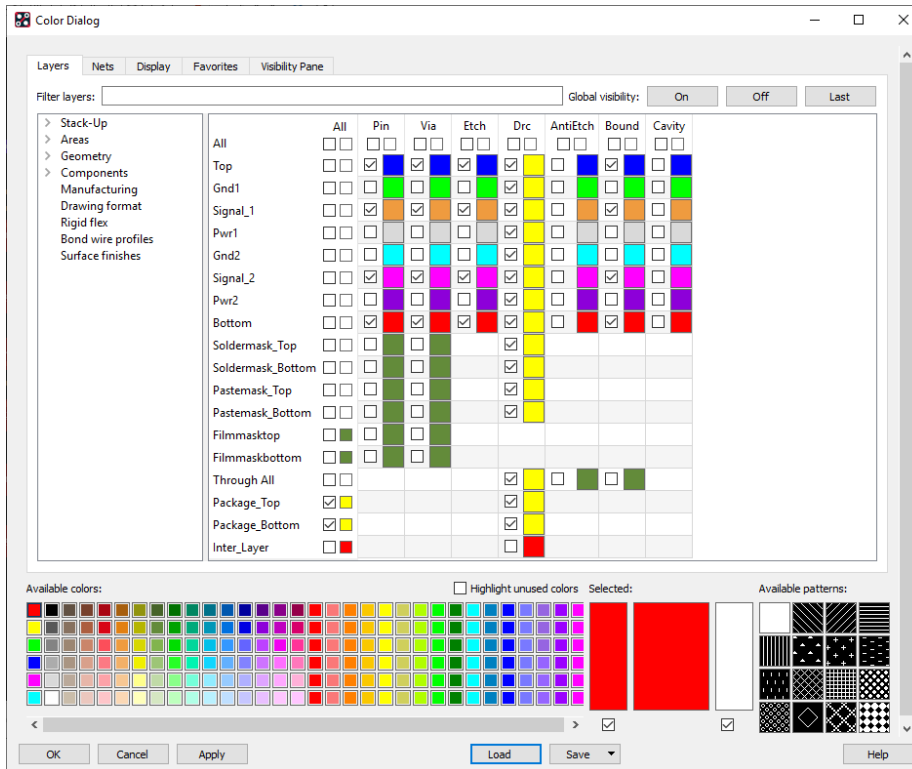
Once the colours have been set create a Colour View file using View>Color View Save (Allegro) or Display > View > Color View Save (OrCAD). The following GUI appears
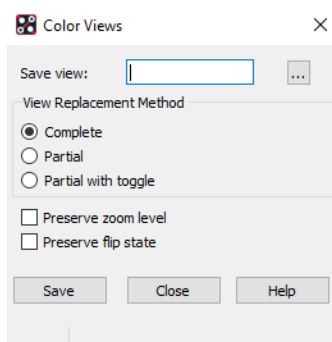
Specify a name for the colour view; the other options are described below.

Complete - Saves the current layer visibility settings to a colour view file. When you load the file later, it completely replaces the design's visibility settings, analogous to how the film option to colour views works.

Partial - Allows a colour view file to be created that stores only *changes* to visibility settings. A partial colour view does not replace all of a design's visibility settings when loaded. It only replaces the items you changed when you created the colour view file. For example, if you changed the colour for all DRCs to be visible, when you loaded that colour 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 colour view file only stores changes. However, settings that you change toggle when you load the colour view file. Toggle means that if the visibility for a layer is on in a design, when you load the colour view file, it is turned off. If it is off, it is turned on.

Preserve zoom level - Includes the zoom points of the current design with the color view.

**Preserve flip state** - Includes the flip status of the current design with the color view.

Then click Save and the colour view file (e.g. Top_Assy.color) is written to the current job directory. Create all the colour view files you require.

There is an option to move any colour view files that you have into a default network location. This way you do not need to copy the colour view files to each job directory to use them. To set this up launch PCB Editor and from the Setup > User Preferences menu locate Paths > Config and define a new entry for viewpath
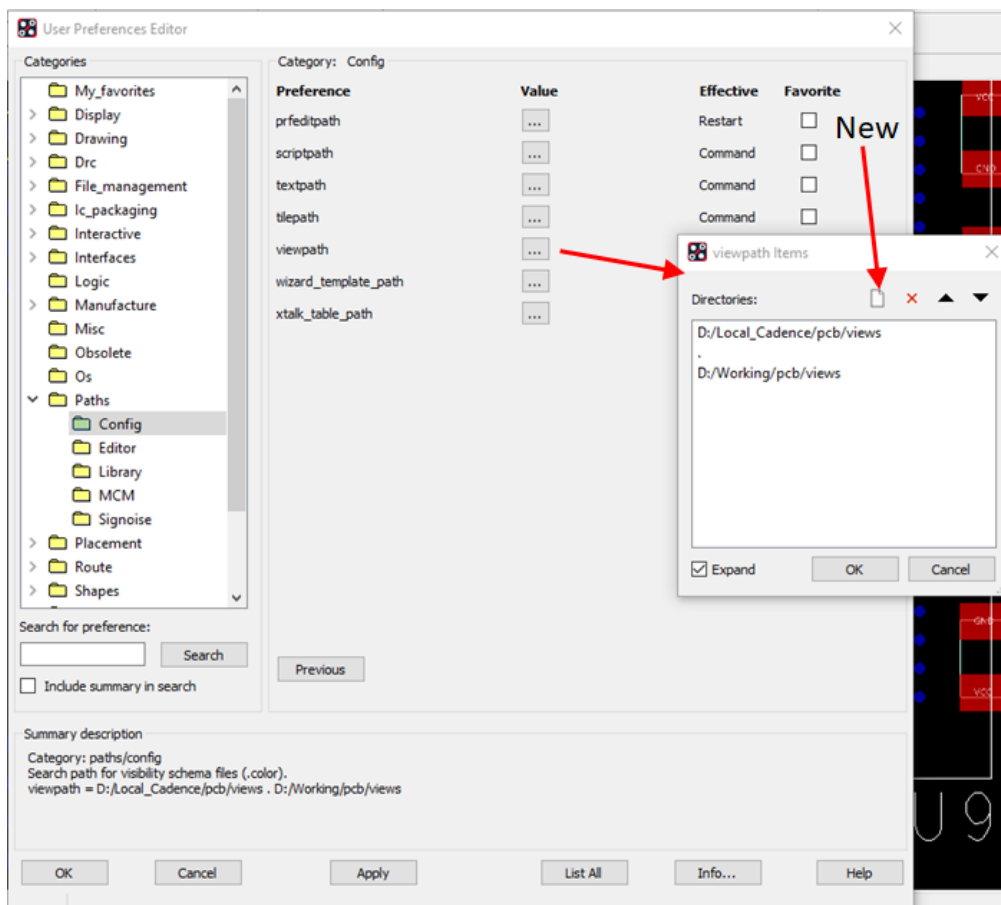
Enter a directory name where all your colour view files will be stored e.g. network_drive\Cadence\pcb\views. Then move all your colour view files to this location. They will be usable once the preference has been saved.

The colour view files are now available from the Visibility tab, click on the Views drop down. The first list is the colour view files; the next group is a list of any Artwork films that are part of the design database. To display these you need to launch the Manufacture>Artwork GUI, add any artworks required, click OK. The artwork films will now be in the list. Every time you now add an artwork it will appear in the list automatically. Select the name of the colour view file and the display will be updated.
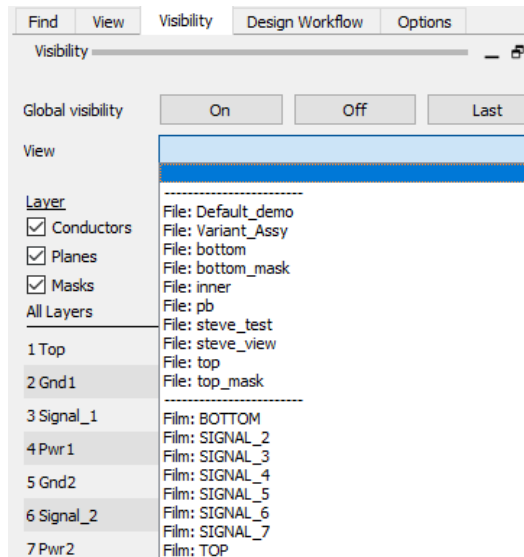

ColorView to Toggle the Placebound Shapes On and Off

During Component Placement, there may be times you want to toggle on/off the Placebound shapes. You can toggle visibility with the Color Views function. Go to View > Color View Save (Allegro) or Display > View > Color View Save (OrCAD) then enter the name of the file you wish to save; "pb" in this example. Select the radio option "Partial with Toggle". This will append the current visibility with the contents in file "pb". It will also allow you to toggle it off.

Note the instruction at the bottom of the form "Change layer visibility". This is your opportunity to open the color dialog and enable the visibility of the Placebound shapes. Use Display > Color/Visibility (Allegro) or Setup > Colors (OrCAD) and turn on the Package Geometry Place_Bound_Top and _Bottom layers.
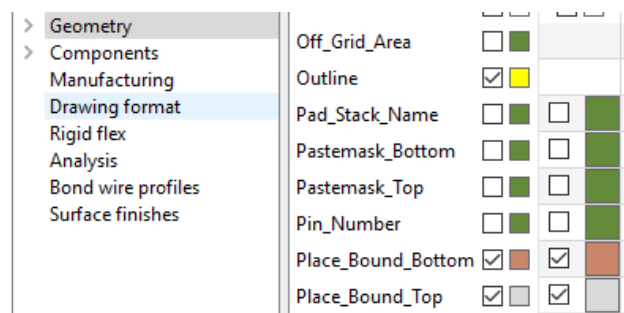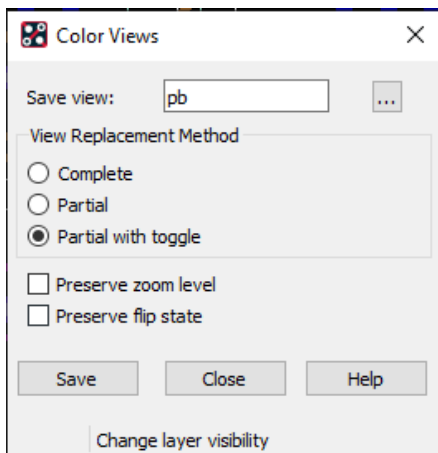
Close the Color dialog then save your color view file "pb" by clicking Save. You will need to move the pb.color file from the current job directory to the views folder defined in the steps above for viewpath. Access the color view file "pb" by making a selection in the "Views" field located at the top of the Visibility Form. Click it again to toggle the display OFF.
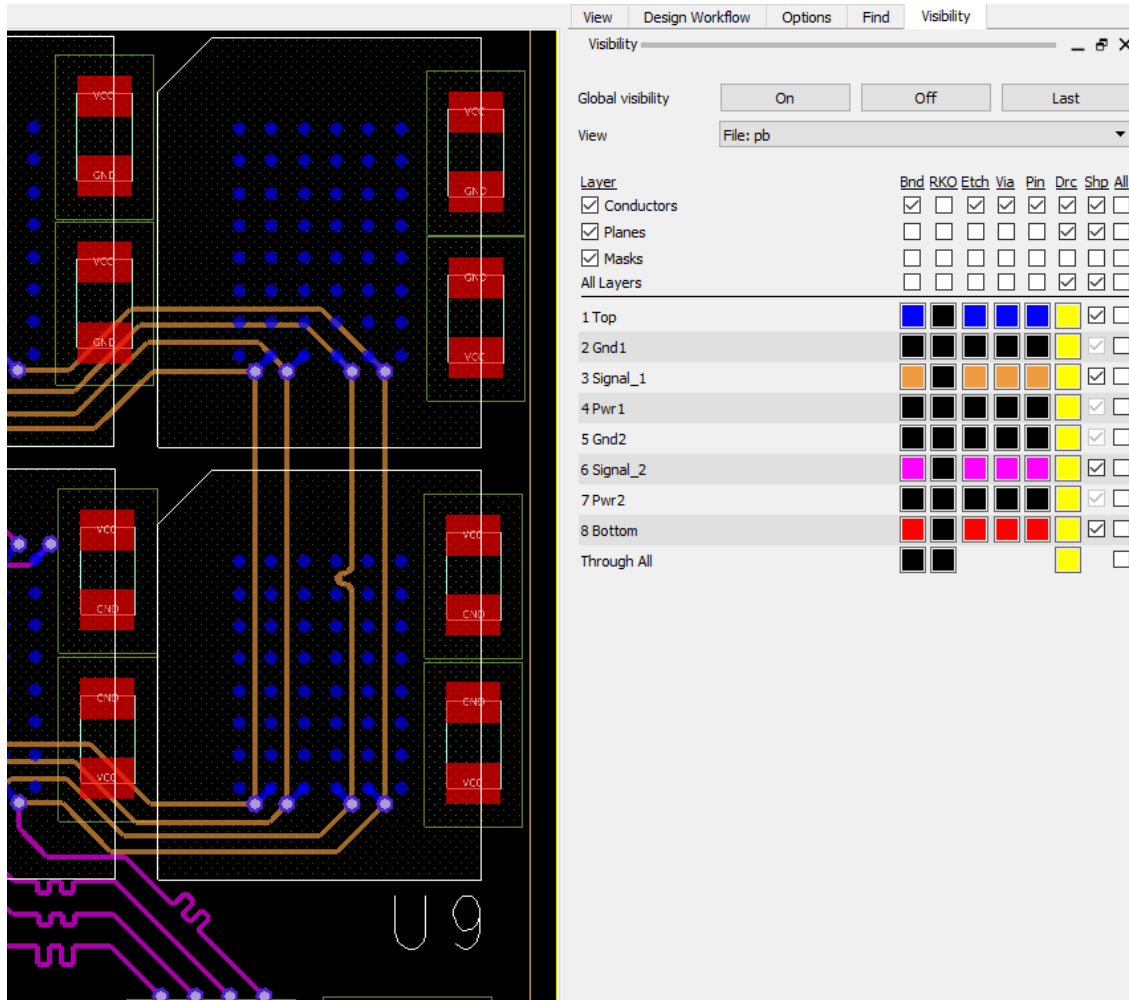
To enable this as a funckey close PCB Editor down and add the following to the env file located in %HOME%\pcbenv folder.

funckey p FORM vf_vis colorview_list File: pb

Re-launch PCB Editor and a key press of p toggles the Placement Boundary layers ON and OFF.