

Title: Panelization with Reuse

Product: Allegro PCB Editor with Performance Option or higher

Summary: The Reuse function can be used to create panels in PCB Editor

Author/Date: Beate Wilke / 07.02.2011

Table of Contents

1	Prepare single Board.....	2
1.1	Export Technology File	2
1.2	Change Shapes	3
2	Create Module.....	3
3	Create Panel	4
4	Update Panel	7
5	Copper Filling	7

1 Prepare single Board

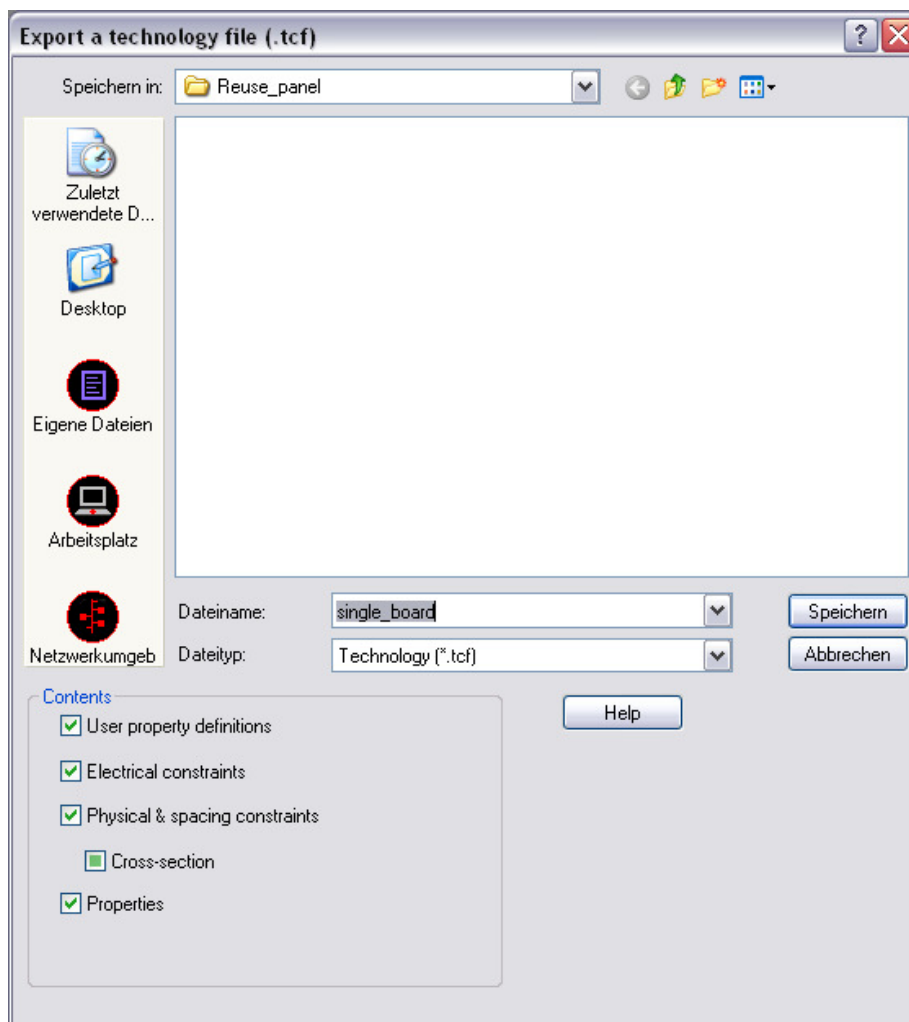


After you finished your board do a save as to a new file name. This is important because you have to do some changes.

1.1 Export Technology File

Export your technology information like spacings, physical setting, layer stackup and other setting into technology file.

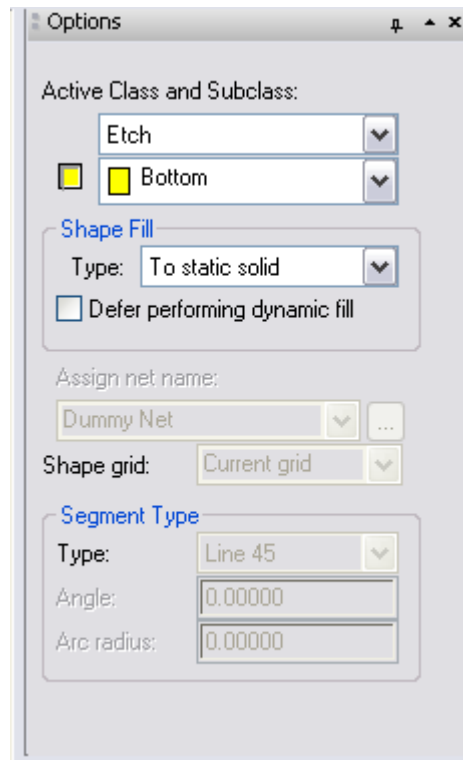
To do this open Constraint Manager and select **File > Export > Technology File**.



Define **Contents** on the lower left corner, put in **file name**, define **path** and select **Save/Speichern**. Save technology file in the local folder. Then it's not necessary to define any path.

1.2 Change Shapes

You need to change all dynamic shapes into static shapes. Go to **Shape > Change Shape Type**.



In Option panel set Type to **To static solid** and select your Shapes.

It's easy to change dynamic shapes to static shape but it's difficult to change static shapes to dynamic. That's the reason why you should work in a copy of your finale board to create panel.

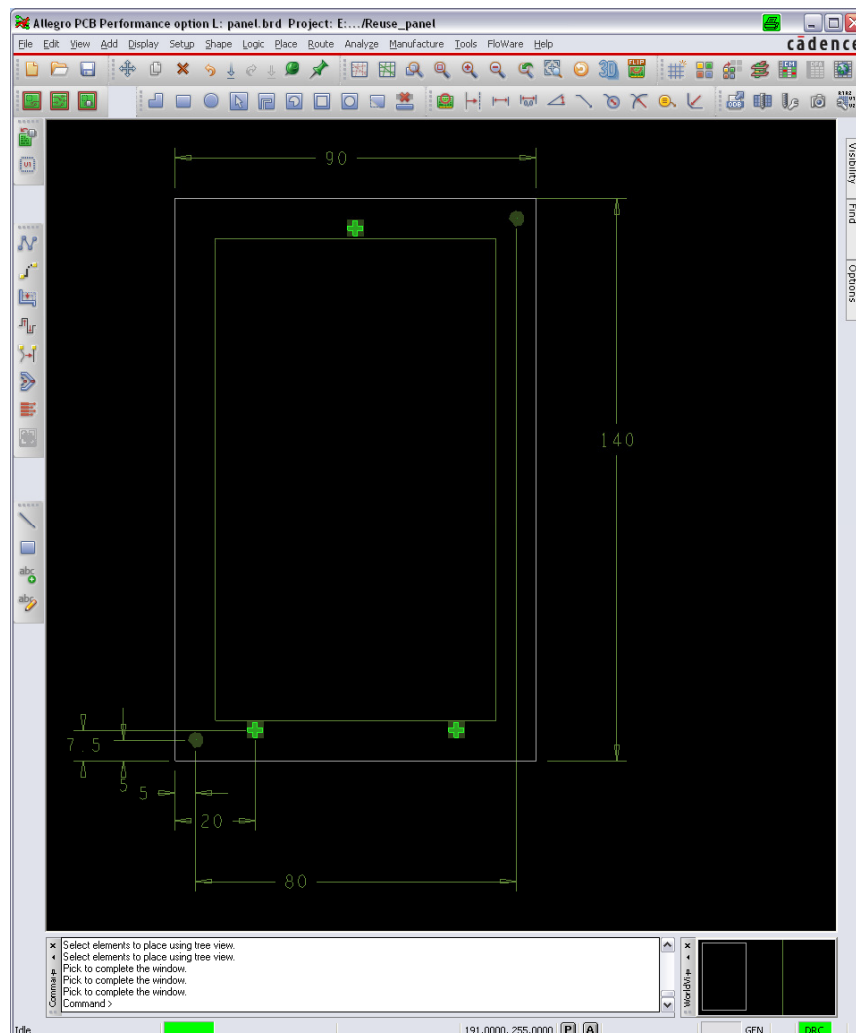
2 Create Module

Turn off all Subclasses. Then turn on only Subclasses you need in panel like conductor layer, mask layer and silkscreen layer. All Keepins and Outline must be turned off.

Go to **Tools > Create Module** and select the whole board and all you need in panel with a box. Next you need to define panel origin with a left mouse button click. Save single board in mdd format into local folder.

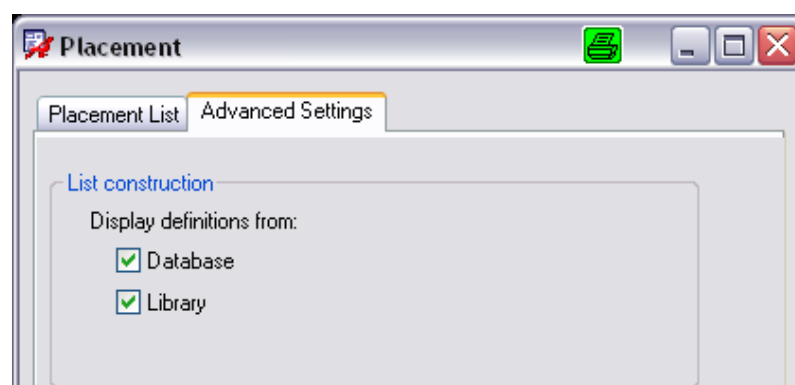
3 Create Panel

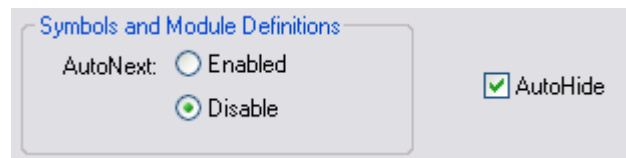
Use **File > New** to create a new board file. Define panel outline, passer and holes. Or use template board with already contain panel definition.



Import technology file from single board. Use **File > Import > Technology File**.

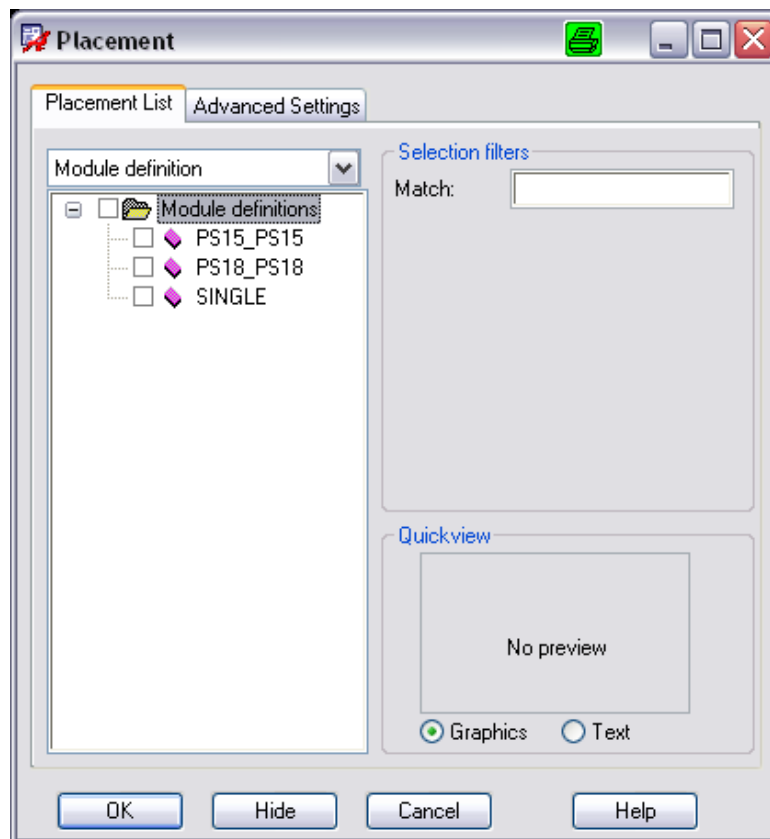
Open **Place > Manually > Advanced Setting** and enable **Database** and **Library** checkbox.



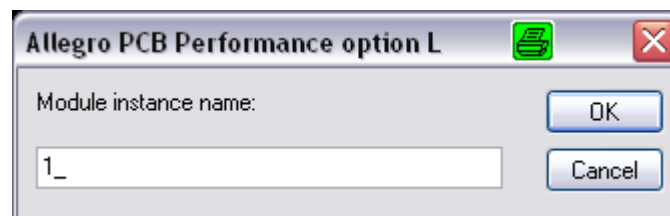


Enable **AutoNext – Disable** and **AutoHide**.

Change to **Placement List** tab and select **Module definition**. The list shows all local and path defines modules.



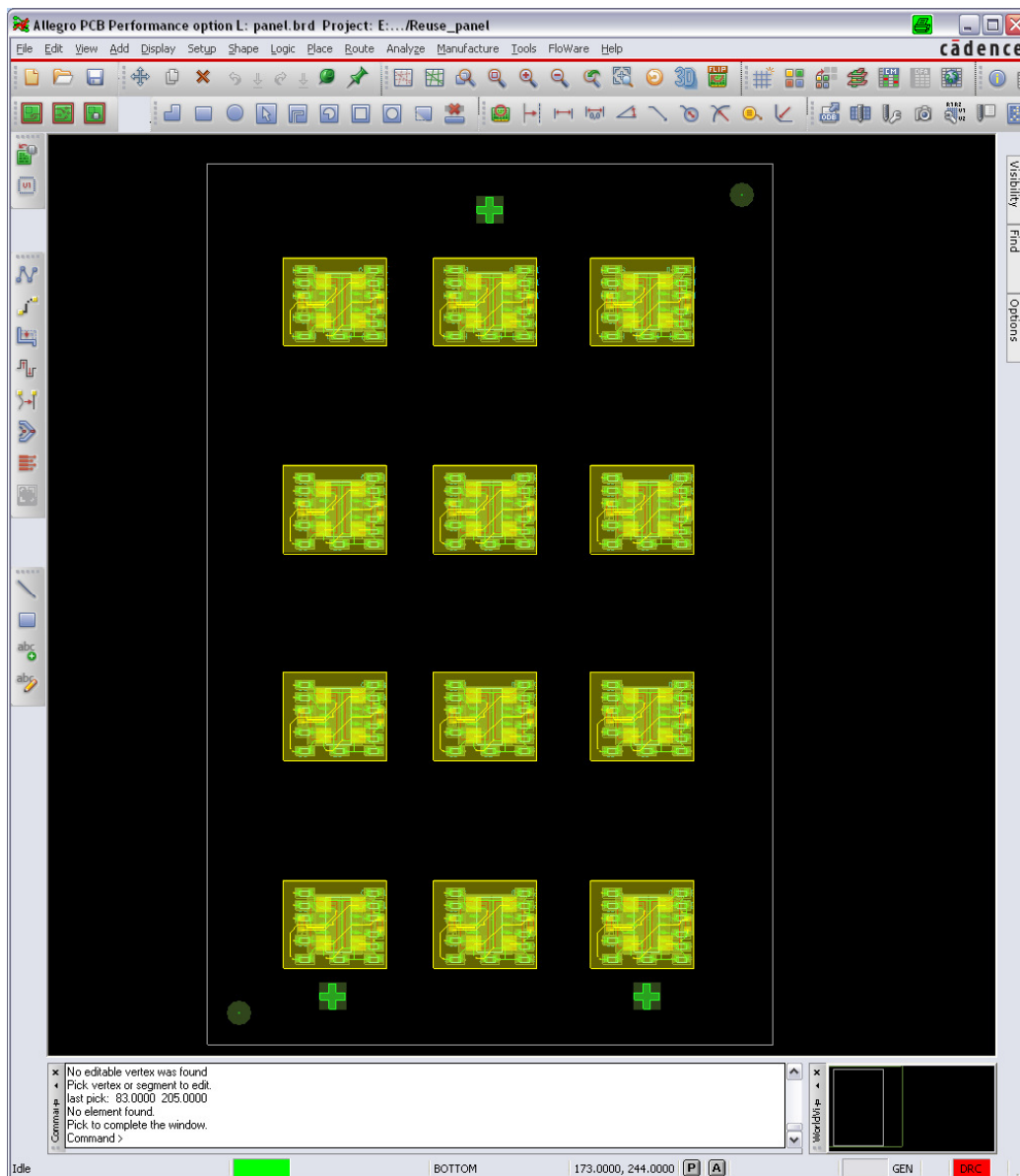
Select the module you want to place and move cursor into graphic window.



You need to define a prefix for every placed module. The prefix is used to get unique net names and part references.

Place several modules in the panel. You can combine different modules in one panel and rotate and mirror them. If you want to move already placed modules set Find Filter to Group to select the modules.

The result can look like this.

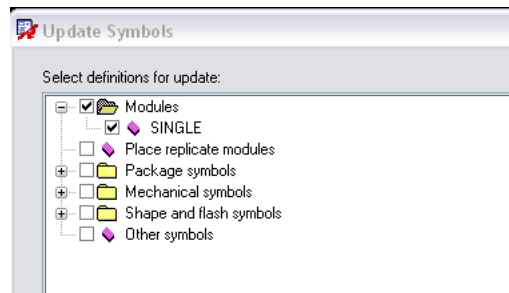


You can use this panel to create complete Gerber, NC drill, pick & place list, netlist, etc. It's an intelligent solution for panels.

4 Update Panel

If you do a redesign you can update your panel and don't need to define a new one. Save the changed board as module under the same file name. Remember the changes you have to do from Chapter 1.

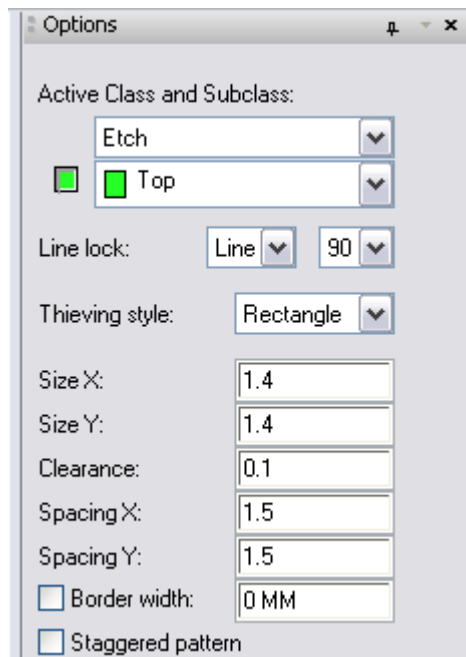
Open panel and do **Place > Update Symbols**. You will find the modules at the top of the list. Enable the checkbox in front of the module name und click refresh.



All changes are updated in the modules.

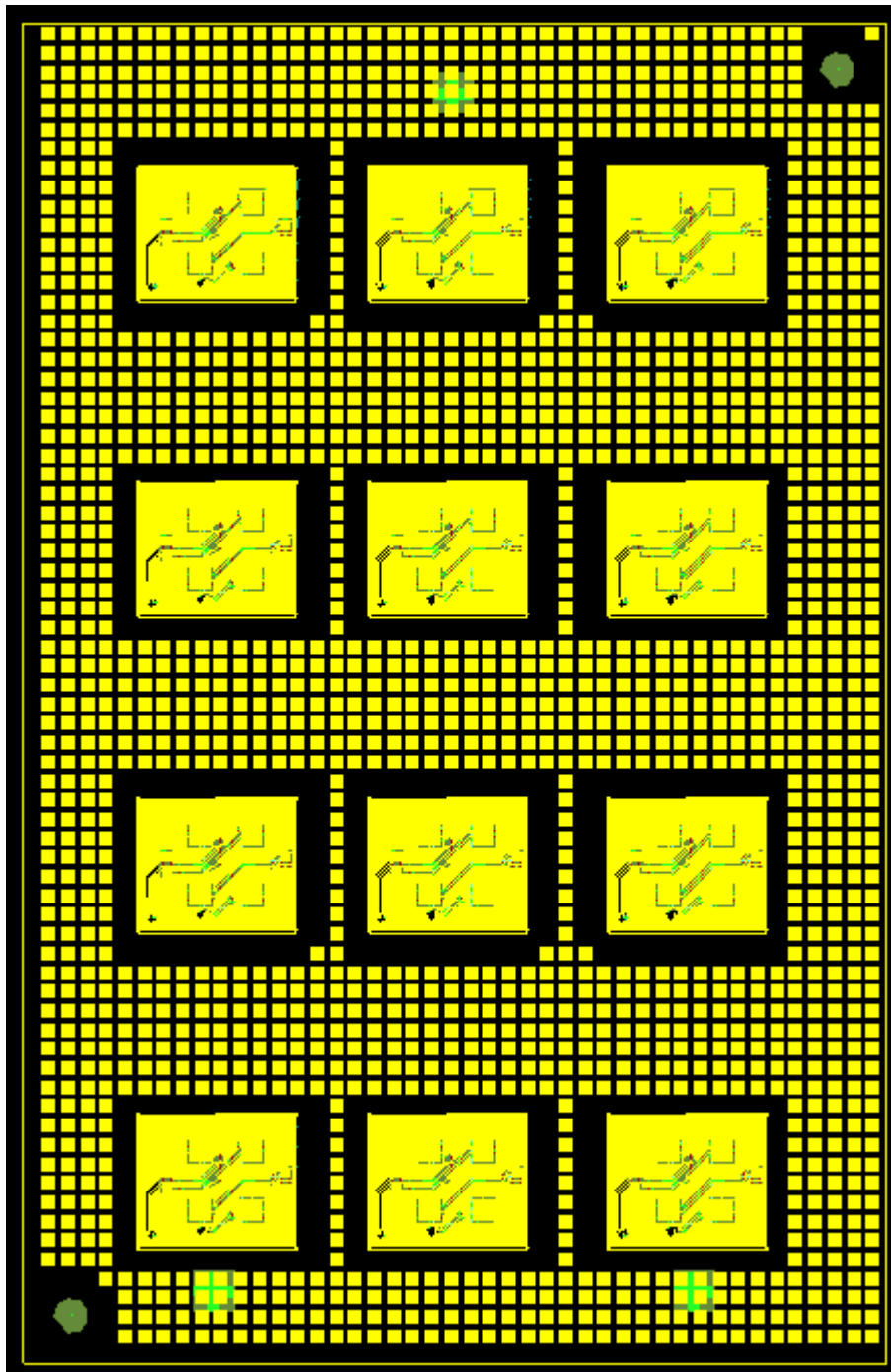
5 Copper Filling

Often the user needs to fill the copper free areas on the panel with copper figures like circles or rectangles. Go to **Manufacture > Thieving** and look into **Option** panel.



- **Line lock** – Setting to draw the thieving border
- **Thieving Style** – Can be changed between Circle and Rectangle copper shapes
- **Size X/Y** – Size of single thieving figures
- **Clearance** – Clearance between copper from the single board and thieving figures
- **Spacing X/Y** – Centre to centre spacing of thieving figures.
- **Border width** – Clearance to drawn thieving border
- **Staggered pattern** – Sets default or staggered style.

Set the setting like you want and draw thieving outline. PCB Editor fills the panel with copper figures following the spacing rules.



The result can look like this. If you want to delete all thieving figures because of wrong parameters, change **Find Filter** to **Group**. All thieving figures from one run are member of the same group. If you want to delete a single thieving figure, change **Find Filter** to **Via**.