

How to Create and Use a Parameter File

## Introduction

PCB Editor has many options for a user defined setup for example colours, text sizes, artworks, Design Parameters like design units, decimal places etc. For most users these settings are the same for design to design or may be driven by a company standard. PCB Editor gives the user the option to export a Parameter File and reuse these settings across multiple designs.

## How to Create a Parameter File.

It is recommended to take settings from a design that has already been setup (or you can start a new design and setup your preferences for colours, artwork films, design parameters, text sizes. Once you have a design setup that is suitable for your requirements use File > Export > Parameters (Allegro) or Export > More > Color/Board Parameters (OrCAD). The following GUI will appear:-

Export Allegro Parameters	Х
Output File Name:	
Available Parameters	
🗌 🛅 Design Setting	^
C C Artwork	
🗌 🧰 Color Layer	
🗌 🧰 Color Palette	
Color Profile	~
Select All Clear All	
Export Close Viewlog Help	

Specify an output filename and check from the available parameters the items you wish to use (this is normally all of them). See below for a description of each setting.

**Available Parameters -** Displays a list of the possible parameter records available for export. If no parameter record exists in the database, it does not display. Click to include the parameter group in the .prm file.

Design Setting - Global values and grid settings.

Artwork - Artwork film definitions.

Color Layer - Priority in which layers are drawn.

**Color Palette -** parameters and colour table.

Color Component - Custom colour for symbol instance of the component

Color Profile - Custom color for wire profile group

**Color Net** - Net custom colour and states. When a file containing net colour data is imported into any design, only the nets that exist in that design are read; the rest are ignored. Net colour assignments are not overwritten, but

rather incremented. To completely replace net colour assignments, click *Clear All Nets* in the *Nets* section of the Colour dialog box before importing a file containing net colour data.

Text Size - Text size settings.

**Application or Command Parameters** - All other supported parameters, including those for auto rename, auto assignment, auto silkscreen, global dynamic fill, autovoid, export logic, drafting, gloss line fattening, gloss dielectric generation, Options window tab settings, test prep, automatic placement, auto swap, thieving, backdrill, interactive flow planner (Allegro only), and Signoise analysis.

To create the Parameter file, click on Export. The file (My\_Parameters.prm) is written to local job directory.

## How to use a Parameter File.

If you have multiple Parameter files there is an option to move any parameter files that you have into a default network location. This way you do not need to copy the parameter files to each job directory or browse to use them. To set this up launch PCB Editor and from the Setup > User Preferences menu locate Paths > Library and define a new entry for parampath

🔀 User Preferences Editor				$\times$	
Categories	Category: Library				
My_favorites	Preference	Value	Effective	Favorite	
> 🛅 Display	devpath		Command		
> Drawing	interfacepath		Command		
> Drc	miscpath		Command		
> Clc_packaging	modulepath		Command		
> 🛅 Interactive	oadoath		Command		
> D Interfaces	paupaun		Command		
	parampath		Command		
Misc	psmpath		Command		
Desolete	step_facet_path		Command	□ New	
Cos Os	step_mapping_path		Command		
✓ ☐ Paths	steppath		Command		
Config	techpath		Command		
Library	topology template path		Command		
С мсм	······································				
C Signoise			🎛 parampath Items	×	
> Desta			Disastasias	1 × 1 ×	
> Koute			Directories:		
· · · · · · · · · · · · · · · · · · ·			D:/Local_Cadence/nch/u	narameter	
Search for preference:			bi/ cocal_cadence/ pcb/	purumeter	
Search					
Include summary in search					
Summary description					
Category: paths/library					
Search path for parameter files (.prm). These allow reuse of physical design data option settings parampath = \$parampath					
			Expand (	OK Cancel	
OK Cancel	Apply	List A	II Info	Help	

To import a Parameter file into a Parameters (OrCAD). The followi	ny board use File > Import > Parameters (Allegro) or Import > Color/Board ng GUI will appear:-
	🔐 Import Parameter File X
	Input parameter file:
	Library
	Import Close Viewlog Help
You can either browse to the Par	ameter file location or if you have defined the parampath above you can click on
the Library button. The following	g GUI then appears:-
	Select Parameter File to Load
	DesignSettings
	my_default
	steve_standard
	Help
	Database
	Library
	Total elements: 4
	it.
Diale the appropriate file aligh on	OK and then Import. The Decemptors are imported. A log file will be evoluble
giving a summary of the paramet	ters imported
Siving a summary of the parameter	
The following are trademarks or registered trade Allegro <sup>®</sup> , Cadence <sup>®</sup> , Cadence logo <sup>™</sup> , Concept <sup>®</sup> , N	emarks of Cadence Design Systems, Inc. 555 River Oaks Parkway, San Jose, CA 95134 NC-Verilog®, OrCAD®, PSpice®, SPECCTRA®, Verilog®
All other trademarks are the exclusive property of	of their prospective owners.
implementation of this feature, application, or st	riging this design, code, or information "as is." By providing the design, code, or information as one possible tandard, Parallel Systems makes no representation that this implementation is free from any claims of
whatsoever with respect to the adequacy of the	any rights you may require for your implementation. Parallel Systems expressly disclaims any warranty implementation, including but not limited to any warranties or representations that this implementation is free
irom claims or intringement and any implied war	

How to Create and Use a Parameter File