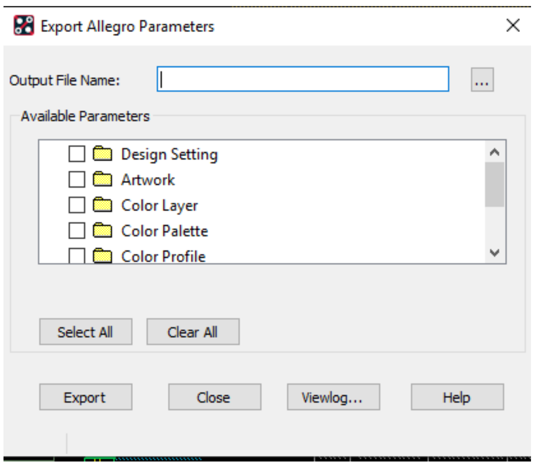
Introduction

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:-



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.

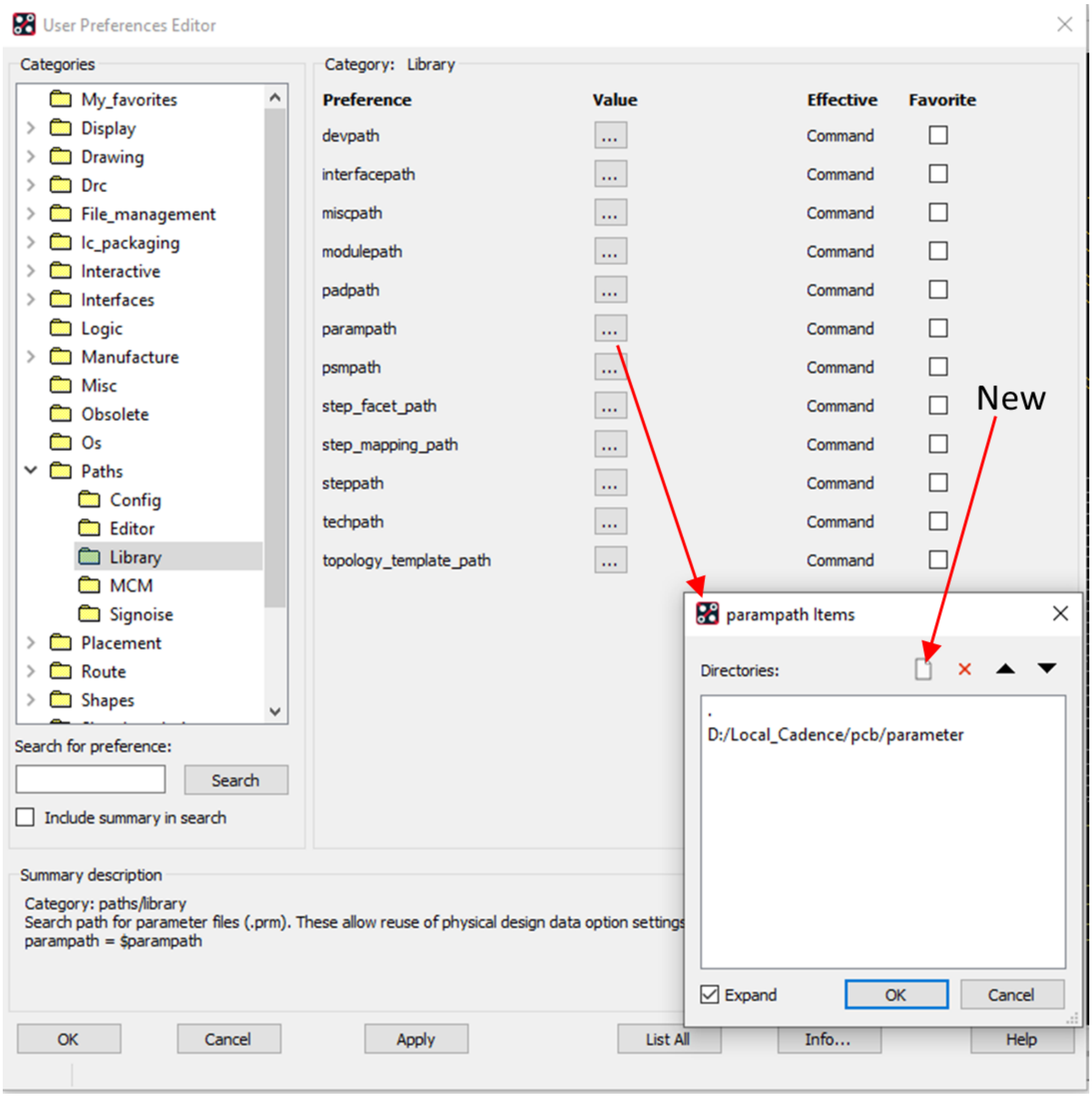
Text Size - Text size settings.

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.

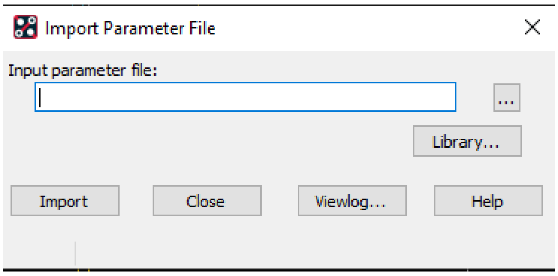
To create the Parameter file, click on Export. The file (My_Parameters.prm) is written to local job directory.

How to use a Parameter File.

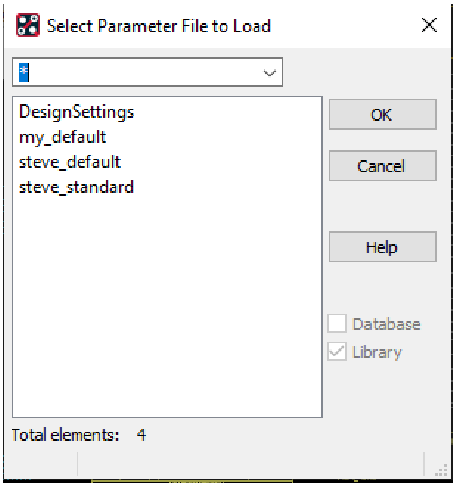
If you have multiple Parameter files there is an option to move any parameter files that you have into a default network location. This way you do not need to copy the parameter files to each job directory or browse to use them. To set this up launch PCB Editor and from the Setup > User Preferences menu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



To import a Parameter file into any board use File > Import > Parameters (Allegro) or Import > Color/Board Parameters (OrCAD). The following GUI will appear:-



You can either browse to the Parameter file location or if you have defined the parampath above you can click on the Library button. The following GUI then appears:-



Pick the appropriate file, click on OK and then Import. The Parameters are imported. A log file will be available giving a summary of the parameters imported.