

User Defined Templates

Start new PCB Editor Designs from a User-defined template

Many users want to use a template board file to seed new designs. First you need to create a template board file. To do this use File>New then specify name and location. Once you start the board you can define default colours (Setup>Colors), rules (Setup>Constraints and Setup>Constraint Modes), default cross sections or layer stackups (Setup>Cross Section), default design Parameters (Setup>Design Parameters). You can also import a default board profile(Import>MCAD>DXF or IDF/x) or create default artwork films (Export>Gerber then Add to add a FILM_SETUP.txt) to name a few. Once the board file is saved there are several ways to use this as a template:

In PCB Editor, File>Open, Browse to a directory holding a "template" board file:

🎛 Open						
Look in:	D:\Local_Cadence\pcb\Templates			• 0 0 0	ß	:: 🔳 P
Desktop	Name	Size	Туре	Date Modified		
Documents	logs signoise.run			19/03/2018 10:58 30/01/2020 12:27		
🚬 steve	💥 SixLayerTemplate.brd			30/01/2020 12:33		
📙 tech						
📒 tech						
📙 Demos						
Customers						
📙 Library						
Local_Cadenc						
< >						
File name:						Open
Files of type:	Board (*.brd)				•	Cancel
Change Directory 🗹						

And open it, then File>Save As the new Design Name in the required location:

💦 Save_As					×
Look in:	D:\Working\ProjectX			- 0 0 0 6	. 📰 🔳 P
Document ^	Name	Size	Туре	Date Modified	
a steve					
📜 tech					
📙 tech					
Demos					
Customers					
Library					
Working					
📕 Local_Cade 🗸					
< >					
File name:	ProjectX				Save
Files of type:	Board (*.brd)			-	Cancel
Change Directory 🗹					

Import>Logic (OrCAD) or File>Import>Logic (Allegro) to load the design netlist data.

Configure the Templates Preference:

Setup>User Preferences, Paths, Config, wizard_template_path, use the "..." button to configure the path, add a new directory for your templates and move it to the top of the list:

		Category: Config			
My_fa	vorites ^	Preference	Value	Effective	Favorite
Display	у	prfeditpath		Restart	
Drawin	ng	scriptpath		Command	
Drc					
	anagement	textpath		Command	
lc_pac		tilepath		Command	
Interace		viewpath		Command	
Logic	ices	wizard_template_path		Command	_
Manuf	facture				
Misc		xtalk_table_path	<u> </u>	Command	
C Obsole	ete		wizard_template_	anth literan	×
🗀 Os			wizard_template_	bath items	^
Paths			Directories:	_ <u> </u>	-
Co Co	onfig				
🗖 Ed			D:/Local_Cadence/p D:/Local_Cadence/p		
🗀 Lib				4/share/pcb/pcb_lib/sym	ıb
C M		Add 🦯			
C Sig		Auu			
Placen	nent	Directory			
Shape					
-	*				
arch for prefe	rence:	_	<		>
	Search	Previous			
and the second se	mary in search		Expand	OK Cancel	

Start a New design and specify a Template:

To start a new design, File>New, Browse to the new location for the design, provide a Name and use the Template button to select a template. (This will support "any" Template name):

💦 New Drawing			×			
Project Directory:	D:/Local_Cadence/pcb/Templa	ites				
Drawing Name:			Browse			
Drawing Type:	Board		Template			
	Board Board (wizard)	🔡 Select Board Ter	nplate			\times
	Module Package symbol	*	~	1	Quickview	
	Package symbol (wizard) Mechanical symbol Format symbol	SixLayerTemplate		ОК		
	Shape symbol Flash symbol			Cancel		
		-		Help		
ОК	Cancel			Database		
				🗹 Library	● Graphics ○ Text	
		Total elements: 1				

© 2020 Parallel Systems Limited

Start a New design and use a default template:

There is a reserved Template Name called "new_default", a template named "new_default.brd" will be used when starting "any" new board design without needing to specify the Template. This Template will be used by **File>New** in PCB Editor and by **PCB>Design Sync** in Capture (CIS) when the PCB Editor "Input Board" entry is blank. The method from Capture (CIS) will create a new design with the netlist data loaded into the template and saved as a new design named after the DSN file (by default)

Note: The Template board in all cases may contain "as much", or "as little", as required to start the design. Some examples of what the Template could contain include colours, units and precision, drawing size and so on, essentially "anything" that does not require a netlist loaded.

("new_default.dra" can be saved in the "wizard_template_path" location(s) to be used as the "starting point" when creating Symbols: Package, Mechanical, Format, Shape or Flash)

The following are trademarks or registered trademarks of Cadence Design Systems, Inc. 555 River Oaks Parkway, San Jose, CA 95134 Allegro®, Cadence®, Cadence logo™, Concept®, NC-Verilog®, OrCAD®, PSpice®, SPECCTRA®, Verilog®

Other Trademarks

All other trademarks are the exclusive property of their prospective owners.

NOTICE OF DISCLAIMER: Parallel Systems is providing this design, code, or information "as is." By providing the design, code, or information as one possible implementation of this feature, application, or standard, Parallel Systems makes no representation that this implementation is free from any claims of infringement. You are responsible for obtaining any rights you may require for your implementation. Parallel Systems expressly disclaims any warranty whatsoever with respect to the adequacy of the implementation, including but not limited to any warranties or representations that this implementation is free from claims of infringement and any implied warranties of merchantability or fitness for a particular purpose.