

PCB Editor Tips and Tricks

Allegro Tips and Tricks

This document provides Tips and Tricks recommended by Ed Hickey, Product Engineering Director of Cadence.

Reducing Mouse Clicks & Travel (Various)

Tips to help reduce the number of mouse clicks & travel to the options panel.

Adding Vias - Use of Space Bar (All Levels)

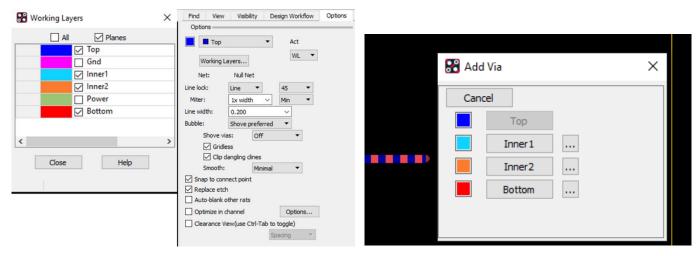
Adding a via has traditionally been done with a double left click of the mouse. The use of the space bar can save you 1000's of mouse clicks per year. I suggest adding the function key below to your local env file. The spacebar entry is represented by "". Other keys can be assigned but space bar is easy to click without looking down.

funckey " " "pop bbdrill -cursor"

Adding Vias - "Working Layer" model (All Levels)

The Working Layer "WL" model to add vias is available in all versions of PCB Editor. It can be used to add conventional through-hole or staggered/stacked HDI vias. Select your target routing layer from a popup GUI that appears adjacent to the via insertion area.

- Order the vias in each Physical CSet
- Click on the "..." to access alternative (least preferred) vias



In Add Connect - Select 'WL"

Select Target Layer during routing

Add Connect – Single click execution (All Levels)

In Etch Edit Application Mode, make a single pick on a pin, via or rat to begin Add Connect. Ensure "Enable Single Pick Execution" mode is enabled. Access this option from the right click – Customize menu.

Add Connect - Add a vertex using a Funckey key (All Levels)

Instead of using a left click to add a vertex point during Add Connect, consider using a Function Key. Simply click the X key every time you want to add a vertex during routing.

funckey x "pick_to_grid -cursor"

Slide – Single Click Execution (All Levels)

In Etch Edit Application Mode, make a single pick on a cline segment to slide it.

Change Active/Alternate Layers using Function Keys (All Levels)

Increment or decrement the active or alternate layer

funckey + subclass -+ (use of "+" character to increment active subclass)

funckey - subclass -- (use of "-" character to decrement active subclass)

funckey a altsubclass -+ (use of "a" character to increment alternate subclass)

Directly change the active layer

funckey 1 options subclass TOP

funckey 2 options subclass SIGNAL_2

funckey 3 options subclass SIGNAL_3

funckey 4 options subclass SIGNAL_4

funckey 5 options subclass SIGNAL_5

funckey 6 options subclass SIGNAL_6

funckey 7 options subclass SIGNAL_7

funckey 8 options subclass BOTTOM

Another method to change the active subclass is from a right click:

Right click > Change Active Layer

Deleting Elements using a Function Key (All Levels)

I think I use this function key more than any other one. Just pass your cursor over a cline, segment, via, text or shape and click "d" to delete it. No click of the mouse!

funckey d "prepopup; pop dyn_option_select @:@Delete"

Moving Components - Single click execution (All Levels)

In Placement Application Mode, make a single left click pick on a component to move it.

Rotating a Component using a Function key (All Levels)

Click the function key R to rotate a component during movement

funckey r iangle 90

Mirror a Component using a Funckey Key (All Levels)

Click the function key M to mirror a component during movement

funckey m "pop mirror"

Snapping using a Funckey Key (All Levels)

While moving an object, use a function key to snap to various elements.

funckey f "prepopup;pop dyn_option_select 'Snap pick to@:@Figure'"

funckey i "prepopup;pop dyn_option_select 'Snap pick to@:@Intersection'"

funckey c "prepopup;pop dyn_option_select 'Snap pick to@:@Arc/Circle Center'"

funckey v "prepopup;pop dyn_option_select 'Snap pick to@:@Via'"

Alias commands to the Middle Mouse Wheel (All Levels)

The "button" command can be used to alias the Middle Mouse Wheel to commands; works with SHIFT, CONTROL and SHIFT-CONTROL combinations.

Examples:

button Swheel_up subclass -+

button Swheel_down altsubclass -+

button Cwheel_up "roam y -\$roamInc"

button Cwheel_down "roam y \$roamInc"

button SCwheel_up "roam x -\$roamInc"

button SCwheel_down "roam x \$roamInc"

In Constraint Manager, those of us with 'older eyes' can use the CTRL + Middle Mouse Wheel to increase the font size.

Find Components or Nets quickly (All Levels)

Use CTRL+F for Components and CTRL+N for nets, then simply type the refdes or netname required. The component or net is highlighted and the item is zoomed to and centred.

alias ~F "prepopup ; pop dyn_option_select 'Selection set@:@Clear all selections';set prompt;prompt 'Find Ref Des';refdes \$prompt;zoom selection"

alias ~N "prepopup ; pop dyn_option_select 'Selection set@:@Clear all selections';set prompt;prompt 'Find Net Name';net \$prompt;zoom selection"

Replacing Padstacks by window selection (All Levels)

In General Edit Application Mode, window around the set of pins or vias you wish to change then use the right click > Replace Padstack > Selected Instances command.

Optional selection window (All Levels)

If a polygon selection window is desired; Right click > Selection Set > Select by Polygon, Lasso, On Path.

Z-Copy – a powerful utility (All Levels)

This is a basic function that everyone should know.

Create a Route Keepin area derived from the board outline

Select Edit Z-Copy(Allegro) or Shape Z-Copy (OrCAD), adjust the options settings to class = ROUTE KEEPIN, Subclass = ALL. Enable 'contract' then enter value the route keepin will be offset from the board outline. Last step is to select the board outline.

Find	View	Visibility	Design Workflow	Options
Optio	ns			
Copy to	Class/Sub	oclass:		
ETCH	ł		•	
TOP			~	
Shape	Options			
	eate dyna	mic shape		
Copy:	Voids			
	Netna	ame		
Size:	Cont	ract		
	🔿 Ехра	nd		
Offse	t: 0.00	0		

Find	View	Visibility	Design Workflow	Options
Option	ns	*12111-000-000-00-000-00		
Copy to	Class/Sub	oclass:		
ETCH	ł		-	
GND			~	
Shape	Options			
Cre	ate dyna	mic shape		
Copy:	Voids			
	Netna	ame		
Size:	Cont	ract		
	🔿 Ехра	nd		
Offse	t: 0.50	0		

Did you know Z-Copy can be used to copy to multiple layers at once?

For example, you wish to create multiple GND planes but the names of each subclass end in GND_5, GND_8, GND_13, etc. Use Z-Copy then in the options tab select one of the GND layers. Edit the numerical part of the layer then enter the wildcard '*' as shown in the graphic below.

Fix/Unfix Elements in the Design (All Levels)

Tips to add and remove the Fixed Property

Quickly unfix all elements

The quickest method to remove the fixed property from all elements in the Design is to:

1. Click the Unfix Icon

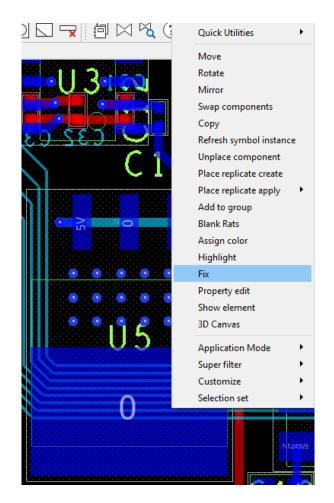


2. Then right click > Unfix All

Done	F6
Oops	F8
Cancel	F9
Persistent select	•
Select by Polygo	on
Select by Lasso	
Select on Path	
Reject	
Unfix All	
Snap pick to	+

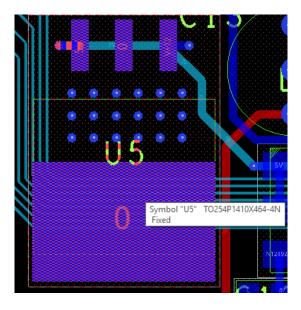
Fix a Symbol's location

Simply hover over the symbol then select the "Fix" command from the context sensitive right click menu.



Datatip display of the Fixed property

You would like to know whether an element is fixed while hovering over it.



Configure the datatip setting as shown below. You may want to enable other symbol related properties to also display in the datatip window. This is done from the Setup > Datatip Customization menu. Select symbol Instance under Object Type then enable the value for "FIXED".

Datatips Customization - Default Settings		×
bbject Type CLine DRC Figure Net Shape Symbol Instance Text Via Via Vid	General Advanced All	Apply Filter
ecify Datatips Format	Origin	■ ■ ✓
Name Symbol name Embedded layer Fixed		Colors: Name and value Value only

Datatip Options (All Levels)

Controls for delay, location and disablement

The User Preference Editor contains hundreds of variables. The "Search" box is a very handy mechanism. In this example, I type in the word "datatip" then click Search. Variables to delay, locate and disable datatips are available to you.

2	User Preferences Editor					×
Ca	tegories	Category: Search results				
,	My_favorites Display	Preference custom_datatip_remove_delay	Value	Effective Command	Favorite	
>	Drawing	datatips_delay		Command		
5		datatips_fixedpos		Command		
>		disable_datatips		Immediate		
Sea dat	Skill Ui Ui Unsupported rch for preference: Indude summary in search					
	mmary description arch for *datatip* found 4 variables.					
	OK Cancel	Apply	List All	Info	н	elp

PCB Editor Tips and Tricks				
The complete set of variables rel	ated to Datatips is locate	ed in the Display > Da	tatips Category	
🚼 User Preferences Editor				×
Categories	Category: Datatips			
My_favorites	Preference	Value	Effective Favorit	e
V 🛅 Display	custom_datatip_remove_delay		Command	
D Alian Cuidan	datatips_delay		Command	
🗖 Align Guides	datatips_fixedpos		Command 🗌	
Datatips	disable_datatips		Immediate	

anywhere

Command

Immediate

Ŧ

Status Form traffic lights (All Levels)

Element

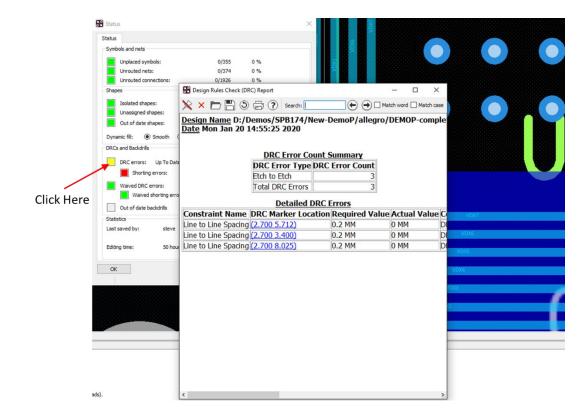
General
Highlight

🛅 Onenal

Display > Status (Allegro) or Check > Design Status (OrCAD) Did you know you can click on the swatch to obtain a report?

disable_hover_over

focus followmouse



"Go to" X,Y Location (All Levels)

Navigate to an X,Y location in the Design

You wish to quickly go to a specific X, Y location in the Design; e.g.: go to location x = 50, y = 50. Adjust your zoom level then make a click on the 'P' button located at the base of the canvas display. Enter in your coordinates then click 'Pick'

Note – Ensure you set application mode = none for this behaviour

PCB Editor Tips						
	器 Zoom (Center		×		
	Type Value Pick	XY Coordinate So 50 Snap to current g Relative (from las Close				
_	48.550,	47.100	mm P A			٩

Disabling Custom Color Assignments (All Levels)

You may inherit a board from another designer or shop and question some of the colour assignments. One quick method to disable colour overrides that may have been applied to nets, buses, Diff Pairs is to go into the colour form. Select the 'Nets' tab on top, you will see a button called 'Hide Custom Colors'. Select that option to revert to 'layer' based colouring.

🎛 Color Dialog						
Layers Nets Display Fa	vorites Visibility Pane	2				
Hide custom colors 💿 Sort as	cending 🔘 Sort descer	nding 🗌	Exclude d	efault nets	s Filter n	ets:
Net group Match group > Bus		Net	Pins	Vias	Clines	Shap
> Diff pair	[XNet] Buf_Out					
> XNet Net	[XNet] Data1.Vd0					

Color Dialog – Open last folder used (All Levels)

Set the variable color_lastgroup located in Setup - User Preferences - UI - Control_panel

🔡 User Preferences Editor						\times
Categories		Category: Control_panel				
🛅 Skill	^	Preference	Value	Effective	Favorite	
Vi		addpin_default_space		Restart		
App_modes		color_lastgroup	\checkmark	Command		
Color		color_nofilmrecord		Command		
Control_panel		color_nosort		Command		
E Fonts						

Display a Layer (All Levels)

Click the color swatch box to the left of the subclass in the Options pane to quickly turn layers on/off during a command. You can also use the Enable Layer select mode in the Visibility tab to quickly turn layers on / off.

PCB Editor Tips and Tricks									
	Find Viev	v Visibility	Design Workflow	Options					
	Options			_					
	🔳 🔳 Тор	•	Act						
	Inne	r2 •	Alt 🔻						
	No availa	able via 🔻	Via						
	Net:	Null Net							
	Line lock:	Line 💌	45 💌						
	Miter:	1x width ~	Min 🔻						
	Line width:	0.200	~			-			
	Bubble:	Shove preferre	ed 🔻			Enable layer select	mode		
	cl								

Invoking Allegro in No OpenGL mode (All Levels)

It may be necessary to run Allegro in a No OpenGL mode; for example, hosting a Net Meeting. Consider creating a Desktop Shortcut for Allegro.exe with the –noopengl option.

PCB Editor 17.4	Properties	:					
Security	Details	Previous Versions					
General	Shortcut	Compatibility					
PCB Editor 17.4 Target type: Application							
Target location: bi	n						
Target: d	ence\SPB_17.4\too	s\bin\allegro.exe -noopeng i					

Display of Properties in the Canvas (All Levels)

Any property attached to an element can be displayed in the canvas. From the Display > Property (Allegro) or Check > Properties (OrCAD) menu, select the Graphics tab then select properties to display from the left column. Clicking the Create button writes out the property values and/or their name to the Class > Manufacturing, Subclass > Properties.

🚼 Show Property				
Information Graphics				
Available Properties	Selected Properties			
3D_Model Adjacent_Layer_Keepout_Above Adjacent_Layer_Keepout_Below Aligned Allow_On_Etch_Subclass	PACKAGE_HEIGHT_MAX PACKAGE_HEIGHT_MIN	● <mark>0 56</mark> ● MM ● <mark>0 56● MM</mark>	CI	1.19
Alt_Symbols Alt_Symbols_Hard Artwork_Prefix Artwork_Suffix Assembly_Constraint_Set Assign_Route_Layer	Subclass: PROPERTIES V	0 <u>5</u> 6° MM 0 <u>56</u> ° MM		
Auto_Rename Autosilk_Use_Oversize Backdrill_Comp_Side_Allowed Backdrill_Exclude	Text block: 1 🗭 🗌 Property Name Text name: 🔍	• (5 6• MM		
Filter Assigning property text to a subclass will delete	Reset Create Delete			
Assigning property text to a subclass will delete	He He		\mathbf{P}	

Suppress Pin-Pin Violations within the same symbol (All Levels)

PCB Designers often use Constraint Regions to suppress pin to pin violations within the same symbol. The property 'nodrc_same_sym_pin' can be applied to the symbol but did you know it can also be applied to the drawing? Applying it at the drawing level has a global impact to all symbols.

- 1. Open the filename.dra
- 2. Edit > Properties > then select Drawing in the Find by Name field.

			View Design Workflow Options Find
🎛 Edit Property		– 🗆 X	Design Object Find Filter
Available Properties	Delete Property	Value	All On All Off Groups Bond wires
Max_Line_Exit_Angle ^ No_Step_Export Nodrc_Sym_Pin_Pastemask Nodrc_Sym_Smap_Pin Nodrc_Sym_Shape_Soldermask Nodrc_Vialist Package_Offset_Bottom Package_Offset_Top Part, Number	Version_Id Pkgdef_Step_File		Comps Shapes Symbols Violds/Cavities Functions Cline segs Nets Other segs Vias DRC errors Fingers Text Clines Ratsnests Lines Rat Ts
Pkg.Pin1_Orientation Pkgdef_Alt_Step_File Name: OK Cancel Help Enter a string in the value field, or leave blank to	Reset Apply Show signore.	2	By Saved Query Find by Query Find by Name Drawing Mare Mare

3. Choose the required Properties on the left, they are added to the right. There are other Nodrc_Sym properties to include if required.

vailable Properties	Delete	Property		Value
Max_Line_Exit_Angle	Nodro	n_Id f_Step_File _Sym_Pin_Pastemask _Sym_Pin_Soldermask _Sym_Same_Pin _Sym_Shape_Soldermask	True True True True True True True True True True True True True	
OK Cancel Help	Reset	Apply Show	•	

4. Click Apply and the properties are added to the Drawing.

Show Properties	_		\times
🗙 🗙 🛅 🖺 🕲 🛱 ? Search: 💽 🕒 🗘	Match wor	d 🗌 Mat	ch case
Symbol D:/Library/Footprints/bgal44c100pl2x12_1300x1300x155.dra	1		
<pre>ARTWORK_PREFIX = bgal44c100pl2xl2_l300xl300xl55_</pre>			
MECH_PIN_TO_CONDUCTOR_SPACING = 0.2032 MM			
MECH_PIN_TO_MECH_PIN_SPACING = 0.2032 MM			
NODRC_SYM_PIN_PASTEMASK			
NODRC_SYM_PIN_SOLDERMASK			
NODRC_SYM_SAME_PIN			
NODRC_SYM_SHAPE_SOLDERMASK			
<pre>PKGDEF_STEP_FILE = BGA144C100P12X12_1300X1300X155.step, 2482</pre>	208, 150	6387908	7, 0,
VERSION ID = 1563879101			
-			

Locked property (All Levels)

The Locked property was designed for Re-Use modules but did you know

The property can be applied to a place replicate circuit to prevent accidental movement of its members. It can also be applied to a package symbol to prevent editing/deletion of the assembly outline or other data elements. Suggestion – consider applying the locked property to the symbol drawing file (at the library level).

🔡 Edit Property		- 🗆 ×
Available Properties Idx_Exclude Library_Path Locked Max_Line_Exit_Angle No_Step_Export Nodrc_Sym_Pin_Pastemask Nodrc_Sym_Pin_Soldermask Nodrc_Sym_Same Pin	Delete Property Locked Version_Id Pkgdef_Step_File	Value
OK Cancel Help Enter a string in the value field, or leave blank to	Reset Apply Show	•

Creating a Command Shortcut with Scriptmode +e (All Levels)

Adding a drawing level property in the previous tip was labour intensive. Let's create a shortcut.

1. Enter scriptmode +e in the command window. This echoes the commands you enter.

Command
trapsize 1336 trapsize 1291 generaledit property edit Adjust find filter then select an element or group of elements. setwindow form.find FORM find name_type Drawing Symbol D:/Library/Footprints/bga144c100p12x12_1300x1300x155.dra has been selected.
Command >

- 2. Copy all the commands above to a single line
- 3. Separate with semi-colons
- 4. Map to a function key
- 5. funckey e "property edit;setwindow form.find;FORM find name_type Drawing;FORM find find_by_name;FORM findname objlist 'Drawing Select';FORM findname done"
- 6. Click the "e" key then press "ENTER" in the command window to launch the property editor dialog!!
- 7. Ideally set this in your env file (located in %HOME%\pcbenv) so this is available in every design.

Symbol D:/Library/Footprin	nts/bga144c100p12x12_1300x1300x155.dra has been selected.
	ct an element or group of elements. hts/bga144c100p12x12_1300x1300x155.dra has been selected. o 1 element(s).
Command >	

Scriptmode +I (All Levels)

"scriptmode +i" is really a shorthand for "scriptmode +invisible"

Let's say you want to create an alias for toggling end cap display but don't want to see the parameter form appear while executing the script.

© 2020 Parallel Systems Limited

alias ec "scriptmode +i; prmed; FORM prmedit display_enhance toggle; FORM prmedit done"

DFA Dynamics - Controlling spacing bubble resistance (Allegro PCB Designer)

Issue - When placing components to DFA rules, it's difficult to pull back the component when the bubble appears.

The resistance can be controlled by the "dfa_pause_level" variable located in Setup > User Preferences > Placement > DFA. Set to 0 for no resistance.

B User Preferences Editor					×
Categories My_favorites Display Drawing Drawing Dre File_management Ic_packaging Interactive Interfaces Logic Manufacture Misc Obsolete Os Paths Diseign_partition	Category: DFA Preference dfa_pause_level display_nodfa_drc_marks	Value	Effective Immediate Immediate	Favorite	

Shape has "No Etch" Status - find it (All Levels)

A dynamic shape is "out of date" BUT it is unfilled.

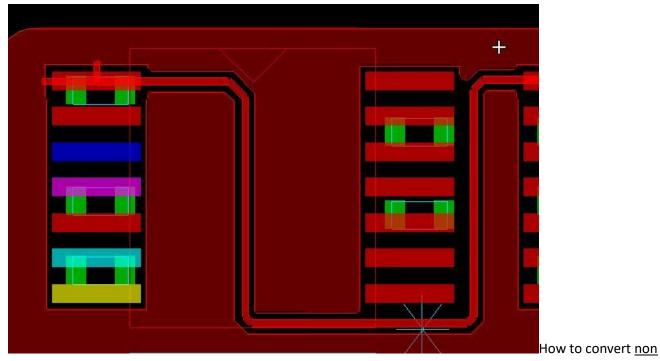
You cannot find the shape in question. It says it's at a particular XY location, on a particular layer, but there is nothing there. Turn on the 'Bound' colour item for that layer to locate the shape outline.

Color Dialog	5 44 5 1° 1532 I A A								
ayers Nets Display	Favorites Visibility F	Pane							
ilter layers:									Glo
> Stack-Up > Areas	All		Pin	Via	Etch	AntiEtch	Bound	Cavity	
 Geometry Components 	Тор								
Manufacturing	Gnd								
Drawing format	Inner1								
Rigid flex Bond wire profiles	Inner2								
Surface finishes	Power								

Voiding of GND clines through GND Plane (All Levels)

A method to auto-void GND sense lines being routed through a GND plane.

In the example below, a GND net is routed through a GND shape. The property 'void_same_net' was applied to the cline. Edit > Properties (Allegro) or Edit > Object Properties (OrCAD) then select the shape and pick the property.



acute angles for shapes to round corners? (All Levels)

Use Shape > Global Dynamic Parameters > Void Controls tab and set the Acute angle control to "Full Round". Static shapes get the same capability by setting the following variable in your env file (%HOME%\pcbenv)

set eap_static_newsmooth env variable.

🎛 Global Dyr	namic Shape Pa	rameters		×
Shape fill	Void controls	Clearances	Thermal relief connec	ts
	rture for gap wid apes less than:	th: O	.102 .635 mm .60403 (sq cm) mm	
Void to su	nd void corner sty uppressed pads (ombined void for ds to hatch grid	le: C gnore dynamic vias added wit	tound tound thamfered uil Round suppression) th Return Path option off	
ОК	Cancel	Apply	Reset	Help

Adding Text from a file (All Levels)

Tip to add text from a .txt file on disk

- Run the add text command: Add > Text
- Set the desired text options on the Options Pane
- Position the cursor with a left click at the location that you want the first line of text to start.
- Right click and choose: Read from File

Find Vie	w Visibility	Design	Workfle	w	Options
Options					
Active Class	and Subclass:				
Board	Geometry	-			
			1		
	ssembly_Notes	•			
Mirror	sembly_Notes	•			
		1.270			
Mirror	size:	1.270			
Mirror Marker	size: :				
Mirror Marker Rotate	size: : ock:	0.000	✓		

Why does the system react slowly when moving a comp? (All levels)

Most likely your large pin-count nets are scheduled Min Tree. Add a VOLTAGE property to large pin-count nets like VCC and GND by Setup > Constraints > Constraint Manager (Allegro) or Setup > Constraints (OrCAD), then on the Properties > Net > General Properties add the required Voltages to the Voltage column.

Is my Database Optimized for Performance? (All Levels)

The performance advisor provides suggestions to increase database performance. The command can be run from Tools > Database Check > Performance Advisor (Allegro) or Check > Database Check > Performance Advisor (OrCAD).

View of file: perf_advisor	-		×
🖉 🗙 🛅 🖺 🕲 🛱 🕐 Search: 🚺 🔶 🔶 🗆 Mi	atch wo	rd 🗌 M	latch case
**************************************	ted.b	rd	^
Ratsnest schedule check			
* OK.			
DRC checks 			
Constraint region check			
<pre>* Region on all - CONN_FLEX: Suggest use by layer or outer/inner regions. ISSUE: Encourage use of new layer based regions which helps performance and simplifies constraint management. shape/region location: (104.150 -20.350) layer: CONSTRAINT REGION/ALL</pre>			
* Region on all - LCD_FLEX: Suggest use by layer or outer/inner regions. ISSUE: Encourage use of new layer based regions which helps performance and simplifies constraint management. shape/region location: (-10.900 181.550) layer: CONSTRAINT REGION/ALL			
* Region on all - BGA: Suggest use by layer or outer/inner regions. ISSUE: Encourage use of new layer based regions which helps performance and simplifies constraint management. shape/region location: (<u>33.147 32.385</u>) layer: CONSTRAINT REGION/ALL			
* Region on all - BGA: Suggest use by layer or outer/inner regions.			M

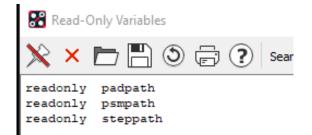
Read Only Variables (All Levels)

A method for a site administrator to make variables read only

Add a readonly entry under the respective variable. In the env file (%HOME%\pcbenv) In the example below, I wish to prevent the end user from modifying PSMPATH, PADPATH and STEPPATH. The command readonly XXXpath is added just below the "set psmpath" variable.

```
set padpath = D:/Library/Padstacks/ symbols . . . ./symbols C:/local_cadence/pcb/padstacks
readonly padpath
set psmpath = D:/Library/Footprints/ . . . ./symbols C:/local_cadence/pcb/symbols C:/Caden
readonly psmpath
set steppath = D:/Library/Step_Files/ D:/Working/cdssetup/OrCAD_Capture/17.4.0/downloaded
readonly steppath
```

To verify in PCB Editor, type readonly at the command line prompt. The window lists variables that are readonly



When an attempt is made to change padpath in the User Preferences editor, the values are greyed out and uneditable.:

Categories		Cat	egory: Library			
My_favorites	^	Pre	ference	Value	Effective	Favorite
> 🛅 Display		dev	path		Command	
> 🛅 Drawing		inte	rfacepath		Command	
> 🛅 Drc		inte	nacepaul		Command	
> Dile_management		miso	path		Command	
> 🛅 lc_packaging		mod	lulepath		Command	
> 🛄 Interactive			path		Command	
> 🛄 Interfaces		pau	paul		Command	
Logic		par	mosth	10-L)	Command	
> 🛅 Manufacture		psn	🔡 padpath Items (Rea	dOnly)		×
Misc		ste	Directories:		ľì × ⊿	_
Obsolete		ste	Directories:			
Os Os		ste	D:/Library/Padstacks/			
Y D Paths		ste	symbols			
Config						
Editor		tec	 /symbols			
🛄 Library		top	C:/local_cadence/pcb/	padstacks		
С МСМ				' share/pcb/pcb_lib/sym	bols	
🛅 Signoise			C:/Cadence/SPB_17.2/	share/pcb/allegrolib/sy	mbols	
> 🛅 Placement			D:/Working/cdssetup/	OrCAD_Capture/17.4.0/	/downloaded_parts/alleg	ropcb
> 🛅 Route						
> D Shapes	~		Expand		OK Ca	ncel
Search for preference:			C CApana		UK Ca	

BBVIA GAP DRC reporting too many DRCs (All Levels)

🔡 User Preferences Editor

Under normal circumstances, Buried/Blind vias separated by less than the minimum bbvia gap rule will generate a DRC no matter how many layers separate them. A drawing level property, BBVIA_SEPARATION, suppresses the DRC if the vias are separated by the specified layer span value or more. In the example below, the property value of "2" suppresses the DRC on the left side. To Set this use Edit > Properties (Allegro) or Edit > Object Properties

(OrCAD) then in the Find by Name dropdown in the Find Pane select Drawing and add the BBVIA_SEPARATION property to the Drawing.

Show Properties	_	\Box ×	
🗙 🗙 🛅 🖺 🕲 📄 🕐 Search:	Match word	Match case	
Drawing D:/Demos/SPB174/New-DemoP/alleg ARTWORK_PREFIX = DEMOP-routed_ ARTWORK_SUFFIX = -ISSUEA CDS_DI_CONTROL_MASK = 0 CLIP_DRAW = CLIP_13 DRC_UNROUTED_MINPROP DRC_UNROUTED_RELPROP	ro/DEMOP-routed.brd		
🔡 Edit Property			~ L
Available Properties 3D_Model Artwork_Prefix Artwork_Suffix Assembly_Constraint_Set Autosilk_Use_Oversize Backdrill_Min_Space Backdrill_Shape_To_Pad Bbvia_Separation Board_Thickness Board_Thickness_Tolerance_Plus Bondfinger_Drc_Disabled Vame: 	Delete Property Same_Net_Spacing_Constraint_Set Spacing_Constraint_Set Physical_Constraint_Set Artwork_Suffix Dyn_Fixed_Therm_Width Room_Type Drc_Unrouted_Relprop Drc_Unrouted_Minprop Bbvia_Separation Bbvia_Separation		
OK Cancel Help	Reset Apply Show	•	

Working with Xnets (Allegro PCB Designer, OrCAD Professional)

Tips for working with Xnets

Create them – Assign default ESpice models to the discrete devices from the Analyze > Model Assignment menu. Resolve / Ignore any warnings then select the Auto Setup button to apply a default ESpice model to the discrete parts.

BBVIA SEPARATION = 2

Note – if the schematic design is Constraint Manager enabled (CM Flow) then xnets are created automatically and the steps above are not required.

🔡 Signal Model Assignment 🛛 🕹 🗙
Devices BondWires RefDesPins Connectors
DevType Value/Refdes Signal Model
 AT91SAM7S32_QFP50P900X900X160-4 AT91SAM7S32 RES_RESC2012X60N_DISCRETE_243 O 243_Ohm DEFAULT_RESISTOR_243.000000OHM_2
< >> Display Filters
Device Type: * Refdes: *
Model Assignment
Model: [OR_243.000000HM_2_1 \rightarrow Auto Setup
Create Model Find Model Edit Model Assignment Map File
Save By Device Load By Device Save By Refdes Load By Refdes

Prevent them on an Instance basis - Assign the property no_xnet_connection at the component level either directly in the board or via a property in the schematic.

Display them Xnets are now available in Constraint Manager. To view you can use Objects – Filter and turn on Nets of Xnet which will list both nets that are part of the xnet.

✓ XNet ✓ Ø Net ✓ Zero Pin		
Voltage	XNet	▲ LONG_SIDE
✓ Nets of XNet	Net	LONG_SIDE
✓ Other	Net	SHORT_SIDE

Naming Convention – The Xnet name defaults to the Net name with the lowest alphanumeric character. In the above example, L is before S.

Change the Xnet Name (Allegro PCB Designer only) – In Constraint Manager, select the Xnet then use RMB – Rename to choose either name of the xnet.

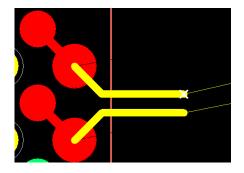
XNet		LONG_SIDE	
Net		LONG_SIDE	
Net		SHORT_SIDE	
Net		VIDCORFI	
Net	🛛 📔 Re	name XNet LONG_SIDE	×
Net			
Net	XNet n	ame:	
Net	SHOR	T_SIDE	~
Net			
Net		OK Cancel	
Net		VUULL	

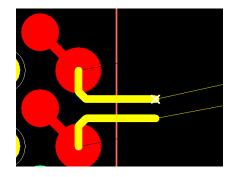
Diff Pair Gathering (All levels)

Looking for tighter coupling at the gathering location?

Decrease the value of the "padentry_factor" variable located in Route – Connect category of the User Preference Editor.





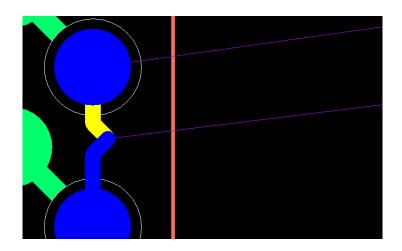


Default

Padentry set to 10

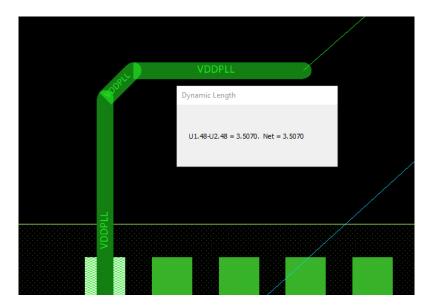
Route a Tandem Diff Pair (All Levels)

Pre route each member as shown in the graphic below then use add connect to route as a tandem pair. This assumes a Diff Pair constraint object has been created.



Display Length Meter for non constrained nets (Allegro PCB Designer, OrCAD Professional)

Set the variable "allegro_etch_length" located in user preferences > Route > Connect



Constraint Manager (Various)

A few tips related to Constraint Manager

Display propagation delay for unrouted nets in CM. (Allegro PCB Designer, OrCAD Professional) Enable the "unrouted" modes in Constraint Manager > Analyze > Analysis Modes > Electrical



Sort results from worst case to best case margin (All Levels)

Double Click Margin Column Header or right click > Sort

Unro L	uted Net Routed/Manhattan Analyze	
*	Sort	
	Display Priority Hide Column	>

Display just the Failed Results (All Levels)

Use Objects - Filter Header and enable Failed only checkbox or simply click on this Icon



Display just nets that have constraints on them? (All Levels)

Use Objects - Filter Header and enable Objects with Directly Set values or simply click on this Icon



Segregate sections of CM with divider lines, i.e. identify the transition between buses and matched groups. (All Levels)

In CM, View > View Options > unsupported and Enable 'Object Type Dividers' or simply click on this Icon

	Objects			Line
Туре	s	Name	Referenced Physical CSet	Min
type		hanc		mm
*	*	*	*	*
Dsn		DEMOP-routed	DEFAULT	0.200
ОТур	8888	Net Classes		
ОТур		▲ Buses		
Bus		DATA[020](19)	DEFAULT	0.200
Bus	888	DATA1[09](10)	DEFAULT	0.200
Bus		DDS[010](11)	DEFAULT	0.200
Bus		MEMORY[029](30)	DEFAULT	0.200
ОТур		▲ Diff Pairs		
DPr	8888	CLOCK	DIFF1	0.127
DPr		D	DIFF1	0.127
DPr		DP_DATA0	DIFF1	0.127
DPr		DP_DATA1	DIFF1	0.127
DPr		DP_DATA2	DIFF1	0.127
DPr		DP_DATA3	DIFF1	0.127
DPr		DP1	DIFF1	0.127
DPr		STDA_SSRX	DIFF1	0.127
DPr	8888	STDA_SSTX	DIFF1	0.127
ОТур	8333	▲ XNets/Nets		
Net		AEN/	DEFAULT	0.200
Net		A16	DEFAULT	0.200
Not	10000	A47	DEFAULT	0 200

Constraint Resolution (All Levels)

You want to see how the constraint resolves between 2 objects?

Use Display > Constraint (Allegro) or Check > Constraints (OrCAD) then window select the 2 elements to obtain spacing resolution; select single element for physical resolution.

In the example below, I window selected the 2 clines of a Diff Pair to obtain spacing resolution. The actual measured gap is supplied in the middle section of the report.

							_	
× 🖻	- 3 6	Seard	h:				⋺ 🗌 Match wor	d 🗌 Mato
		E1 1 4		<u>Constrai</u>	nt Hierarch			
Decemintic		Element 1		mant "Di I	Theb /Dettern!	Element 2	nent "D Etc	h /Delte
Descriptio Location	n			nent D+, i	Elch/Bollom	Odd-angle Line Segr	nent D-, Etc	η/Βοιιο
NetClass		<u>(88.136 53</u>	. <u>504)</u>			<u>(87.424 53.993)</u>		
		1						
NetGroup Bus		1						
bus DiffPair		D				D		
XNet						<u>D</u>		
Net		D+				D-		
PinPair								
NetClass-	NotClass							
Region	NetClass	1						
Region								
		Air gan	distance	hetween el	ements = 0.1	27000 millimeters		
		Air gap	distance	between ele	ements = 0.1	27000 millimeters		
		5.						
	Resolve	5.	solved s	Spacing a	nd Diffpair	.27000 millimeters <u>Constraints</u>	Value	
	Resolve Design	Re d Level So	solved s	Spacing a me Constr	nd Diffpair	<u>Constraints</u>	Value	
		Re d Level So	esolved s urce Nai FAULT	Spacing a me Constr Line to	nd Diffpair aint	<u>Constraints</u>		
	Design	Re d Level Sou	esolved surce Nai FAULT	Spacing a me Constr Line to Differer	n d Diffpair T aint Line Spacing	Constraints Inary Gap	0.2 MM	
	Design DiffPair	Re d Level Sou DEI	esolved s urce Nar FAULT F1 F1	Spacing a me Constr Line to Differer Differer	nd Diffpair raint Line Spacing Itial Pair Prin Itial Pair Nec	Constraints Inary Gap	0.2 MM 0.127 MM	
	Design DiffPair DiffPair	Re d Level Sou DE DIF	esolved S urce Nai FAULT F1 F1 F1 F1	Spacing a me Constr Line to Differer Differer Differer	nd Diffpair Taint Line Spacing Itial Pair Prin Itial Pair Nec Itial Pair Cou	Constraints I nary Gap k Gap	0.2 MM 0.127 MM 0.117 MM 0.01 MM	

Preventing Shorts (All Levels)

- ✓ In PCB Editor, It is possible to override a DRC with properties, make sure that the nets involved don't have the "NO_DRC" property on a pin(s).
- ✓ Make sure there are no "waived DRCs" that are not valid.
- ✓ Run DBDoctor to keep the design in sync and up to date.
- ✓ Make sure the plane layers in the cross section form agree with the artwork control form (negative or positive).
- ✓ Make sure the padstacks are designed properly; both Antipads and thermals
- ✓ Include the IPC-D-356 netlist in your Fab Package; insist the Fabricator does a netlist to Gerber compare.

Suppress SIGNOISE Warnings and Error messages (Allegro PCB Designer, OrCAD Professional)

To suppress Signoise warning and error messages from appearing in the message window, set the variable in the env file.

set sigsuppress ERROR WARNING

Write vs Save (All Levels)

You want to save the current board file you are working on as a new name but want to keep the current file open using its current name. The "Save As" function will make the saved file the active file.

Enter "write" in the command window followed by a design name.

PCB Editor Tips and Tricks	
	Command Select one object to resolve physical constraints, or select two objects to resolve spacing constrative write backup.brd Performing a partial design check before saving. Writing design to disk. 'backup.brd' saved to disk. 'backup.brd' saved to disk. 'b;/Demos/SPB174/New-DemoP/allegro/backup.brd' saved to disk. Closed connection with Capture. > Sending response DoneOpenBoard Opened connection with Capture.
[Command >
Autosave the Databa Setup > User Preferences >	se (All Levels) File_management > Autosave. Variables to autosave the database
🔐 User Preferences Edi	itor

Categories		Category: Autosave			
My_favorites	^	Preference	Value	Effective	Favorite
> 🧰 Display		autosave		Restart	
> 🛅 Drawing > 🫅 Drc		autosave_dbcheck		Restart	
 File_management 		autosave_name		Restart	
🛅 Autosave		autosave time	30	Restart	
Dournals					

Reopen command (All Levels)

Most people do not know about this convenient command which does what is says, reopens a design file.

Enter "reopen" in the command window to reopen the same file.

Command
Opened connection with Capture. reopen Closed connection with Capture. Loading consmgr.cxt Loading acns_formula.cxt Opening existing design > Sending response DoneOpenBoard Opened connection with Capture. (SPMIHDB-213): DRC set to "out of date". The current product selec
Command >

File Management - artwork, plots, log files, reports (All Levels)

Looking to manage your artwork, plots and reports more efficiently? There is a suite of variables beginning with ADS designed to create subdirectories for common data files. Setup > User Preferences > File_management > Output_dir.

Categories	Category: Output_dir			
My_favorites	Preference	Value	Effective	Favorite
> 🧰 Display	ads_sdart	output	Immediate	
> 🛅 Drawing	ads_sdlog		Restart	
> 🛅 Drc	uus_sulog		researc	
✓ ☐ File_management	ads_sdmcad		Immediate	
🛅 Autosave	ads_sdplot		Immediate	
Dournals				
Miscellaneous	ads_sdreport		Immediate	
Dutput_dir	dump_library_directory		Command	
2				

Open File Manager from Allegro (All Levels)

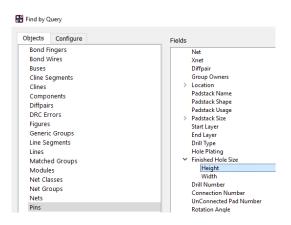
Quick method to explore to your working directory

Tools > Utilities > File Manager (Allegro) or Tools > More > File Manager (OrCAD).

Looking for a particular padstack based on hole size? (All Levels)

There are 2 methods to obtain a report of library padstacks:

- 1. Manufacturing > NC > Drill Customization > Library Drill Report (Allegro) or Manufacture > Customise Drill Table > Library Drill Report (OrCAD).
- 2. Consider the Find by Query command available via the Find Pane. Set up a query to look for a particular Pin / via with a Finished hole size...



Testability - Multiple Probe-Type Support (All Levels)

The Testprep parameter form, Manufacture > Testprep > Parameters (Allegro) or Manufacture > Test points > Parameters (OrCAD) supports entries for multiple probe-types. The names (Probe Type) and centre to centre spacing (Probe Spacing) is user definable.

To add a row in the form, select a Probe Type cell then right click > Add.

ТО	🎛 Testprep Paramet	ers				×
	General Parameters	Padstack Selections	Probe Types			
TP47■''	Enable	Probe Type	Prol	be Spacing	Figure	
			508		Square	V
			Add		Hexagon X	T
			Delete	0.127	Octagon	¥
_ •						
			Re-sort			
P163						
	NOTE: 'Cross' figure wi	II has some of the same have the		Load	Save	
	NOTE: Cross figure wi	li be used for probes th	at are too close.	Load	Save	
$T \cap A \cap \cap$						
	ОК	Cancel			ł	Help
		N70364 N68133	Ρ	26	2	

Drill Legend Support for INCHES and METRIC (All Levels)

Update your default-units.dlt file as shown below (located normally %CDSROOT%\share\pcb\text\nclegend)

```
?AlternateUnits "millimeters"
?ColumnDefinitions '(
           ("Figure"
                       "FIGURE"
                                   7)
           ("Holesize" "SIZE"
                                  15)
                     ("Tolerance" "Tolerance"
                                               15)
                     ("Holesize2" "SIZE MM"
                                               15)
           ("Tolerance2" "TOLERANCE MM" 15)
           ("PlateStatus" "PLATED"
                                     10)
           ("NonStandard" "NONSTANDARD" 15)
           ("Quantity" "QTY"
                                   6)
          )
```

Uprev Symbols (All Levels)

The command "uprev_overwrite" can be used to uprev library files to the current software revision.

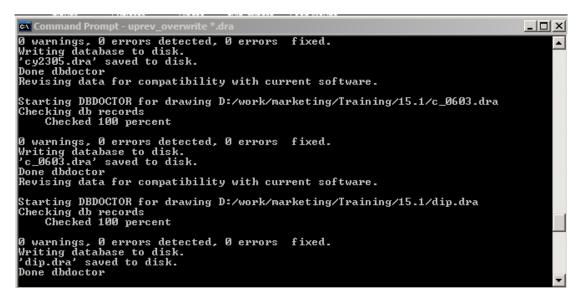
Syntax – uprev_overwrite n where n = *.dra; *.psm; *.ssm, etc

Ex – in a command window, enter uprev_overwrite *.dra to update all .dra files to the current software level.

For more flexibility in upreving in a DOS command window do a:

uprev -help

** Be sure to make a backup copy of your library before uprev



Troubleshooting - Running Allegro in Safe Mode (All Levels)

Many Allegro programs support the "-safe" command line option. This starts the program without any user customizations or extensions. This can be used a debug mechanism to eliminate if user configurations are the cause of the problem with the product.

It disables loading at start-up

© 2020 Parallel Systems Limited

- local env file (%HOME&/pcbenv/env)
- cds_site configuration data
- any user skill code
- pre-register scripts
- ini file which stores window size/position information
- most recent used files (MRU)
- remembered Windows positions (.geo files)

In addition, graphic programs support the "-noopengl" option which will disable the enhanced Graphics based upon OpenGL.

Example: Run an "out of the box" allegro without opengl

allegro -safe -noopengl

Tips from Allegro PCB Users (All Levels)

Join the Cadence Community Website and see what other users are saying

https://community.cadence.com/cadence_technology_forums/f/pcb-design

a d e n c e°	Products Solutions	Services	Support	Community
e:Community:Forums:PCB Design	rums			
All Forums	Q Community Search			
Custom IC Design				
Custom IC SKILL	Toolar	Deplies	All recent discussion	By last post date Cast Post
Digital Implementation	Topics	Replies	Views	
Functional Verification	bottom symbol Pin number not visible in auto cad while exporting DXF file from		243	By kabalee 20 Jan 2020 11:21 PM
Functional Verification Shared Code	allegro ?? started by kabalee on 4 Jan 2020 9:16 PM			
Hardware/Software Co- Development, Verification and Integration	NPTH HOLE NOT VISIBLE IN 3D VIEW (ORCAD PCB EDITOR17.X) started by GK MN on 20 Jan 2020 10:23 PM	0	32	By GK MN 20 Jan 2020 10:23 PM
High-Level Synthesis	VARIANT BOM	0	32	By Araugel 20 Jan 2020 9:00 PM
	started by Araugel on 20 Jan 2020 9:00 PM			

The following are trademarks or registered trademarks of Cadence Design Systems, Inc. 555 River Oaks Parkway, San Jose, CA 95134 Allegro[®], Cadence[®], 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.