cadence®

Allegro PCB Editor: Tips and Tricks

Product Version: SPB 17.2
November 2020

Contents

Purpose

This document provides tips and tricks for using Allegro PCB Editor. It also covers usage and examples to customize the environment to maximize productivity.

Audience

This document is intended for Allegro PCB Editor users who wish to maximize the performance of Allegro PCB Editor for their environment. It also contains FAQs.

Use the feedback button on Cadence Online Support if you have tips or tricks that you would like to contribute.

What SW version was my database saved in?

As of release 17.2 S058, the database stores the last program and release the database was saved with. Use the command dbstat -I to retrieve this information. For the near term, this option is undocumented.

```
C:\>dbstat -I myboard.brd
 myboard.brd:    allegro 17.2 S058 - 8/6/2019
```

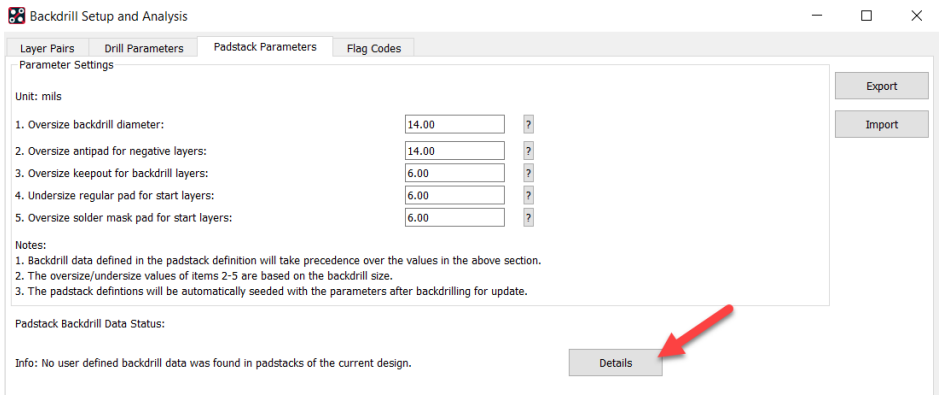
```
C:\>dbstat -I mypackage.mcm
 mypackage.mcm:    apd 17.2 S059 - 9/10/2019

C:\>
```

Maintaining Custom Menus and Shortcuts

Questions related to custom menus, especially around the time of new release migration comes up often. It is recommended to go through Adding and Appending Allegro Menu Items using SKILL.

Backdrill

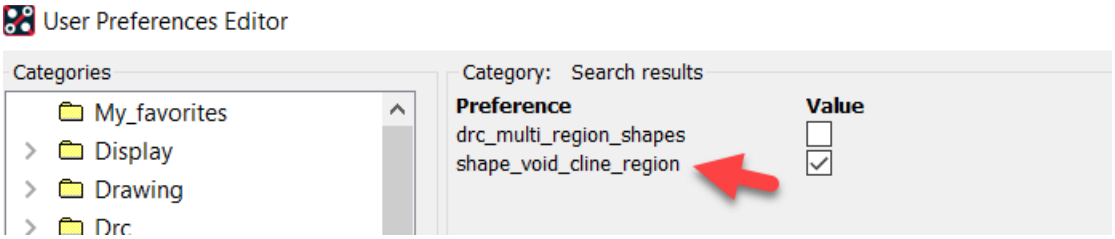
Backdrill can be driven by library data or from the associated parameter form. It's important to know the library driven model will always take priority over the parameter entries. How do you know if all via and pin pads in your database have backdrill library data? Click the "Details" button as shown below for an assessment. This should be done early on to give the library folks time to make the updates.



Constraint Regions – Shape Voiding

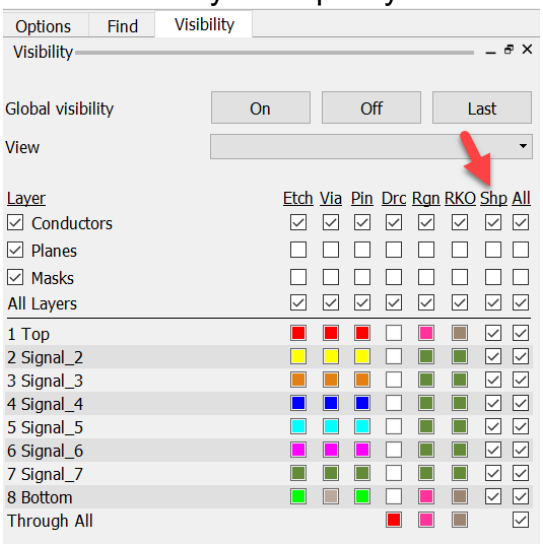
By default, a cline crossing a region boundary will be voided to the larger of the inside/outside constraint values. Customers desire what is referred to as vectorized behavior where the line to shape spacing follows the explicit rules defined. The user preference variable “shape_void_cline_region” can be enabled to change the default behavior.

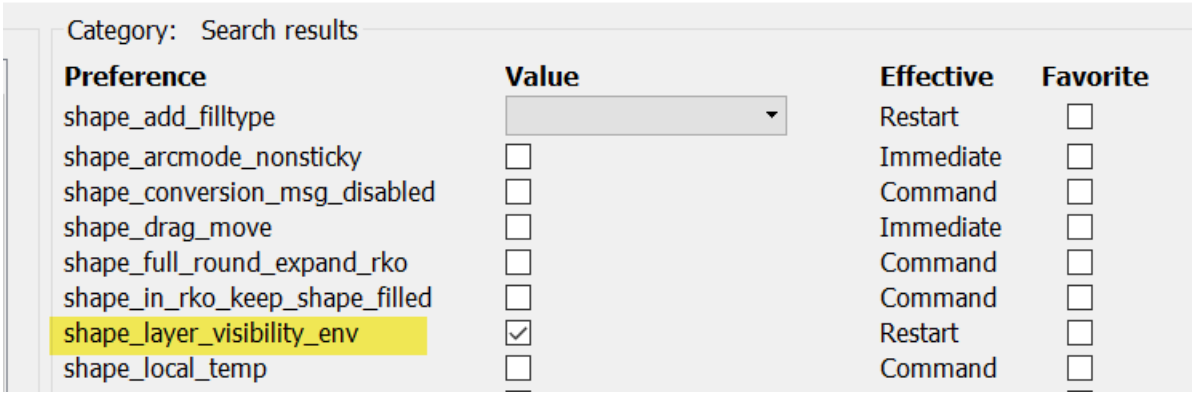
** Please test out the solution first keeping an eye on potential performance degradation.



Add Shape Header to Visibility Pane

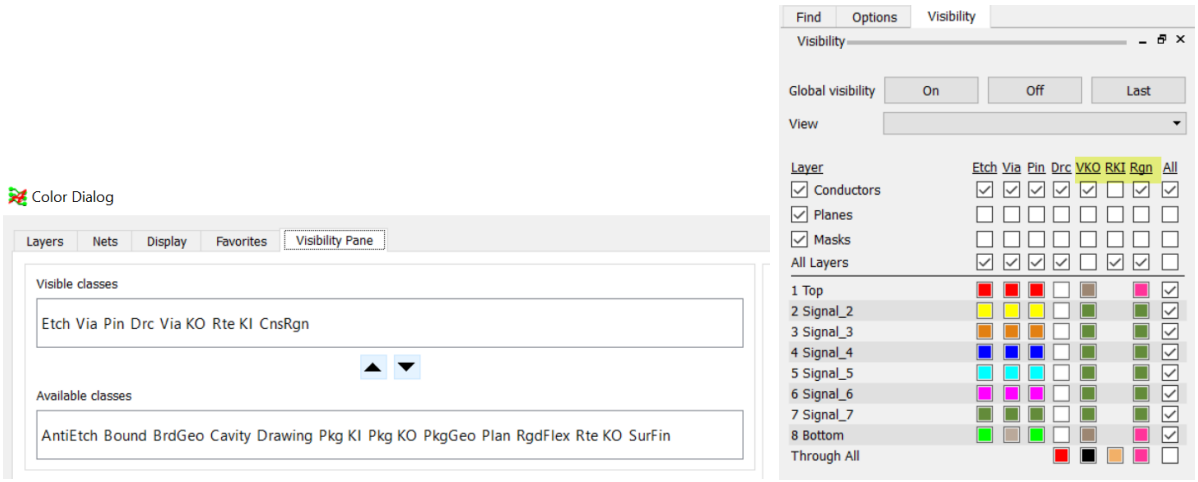
Enable the user preference variable “shape_layer_visibility_env” then restart Allegro. This will allow you to quickly turn on/off the visibility of shapes by layer or globally.



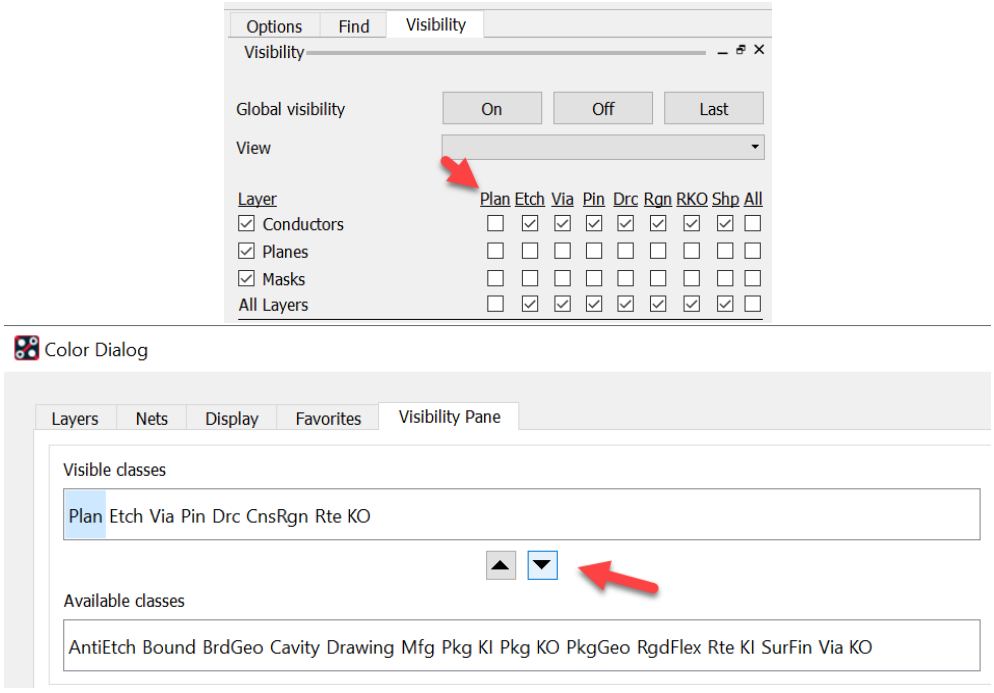


Add/Remove Classes to the Visibility Pane

Add your most frequented classes to the visibility pane by invoking the color dialog – Visibility Pane Tab then drag or double click from the “available classes’ section. In the example below, Via KO, Route KO and Constraint Region classes were appended to the pane header. They are abbreviated to conserve space.

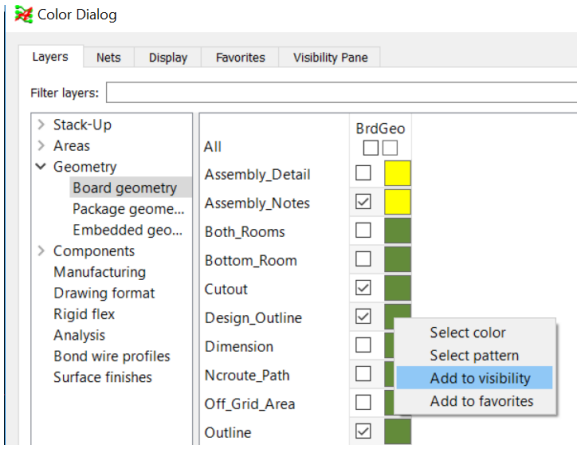


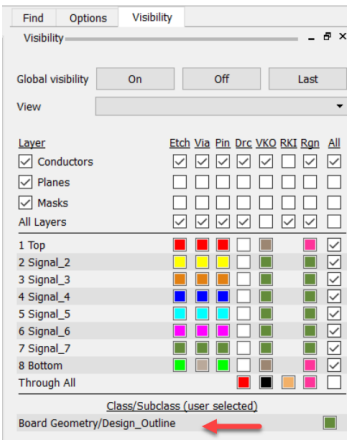
If you do not leverage the flowplan/bundle features, you may wish to remove the "Plan" column. Following the steps provided above, select "Plan" for the "visible classes" section then click the down arrow to remove it.



Add subclasses to the Visibility Pane

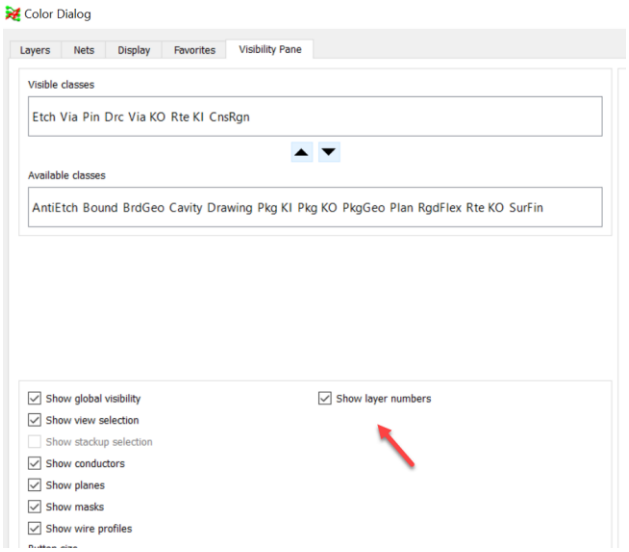
Add your most frequented subclasses to the visibility pane by invoking the color dialog – Layers tab. Navigate to the subclass then hover over the color swatch followed by a "RMB – Add to Visibility". In the example below, "Design Outline" is added to the Class/Subclass section of the pane.

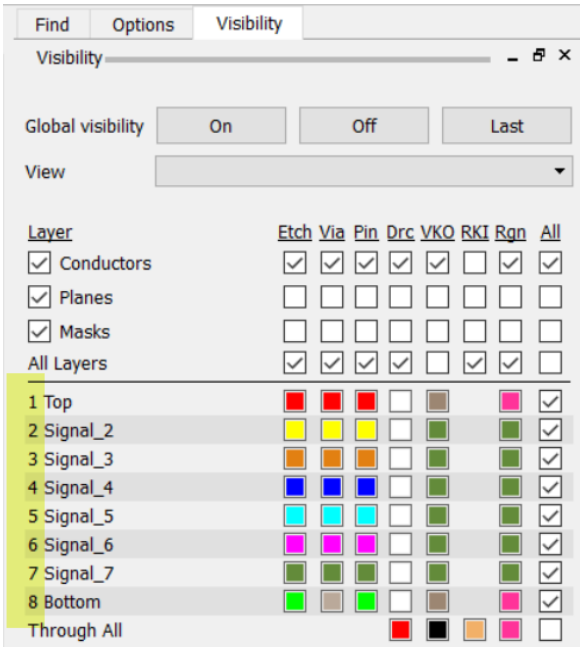




Display Layer Numerical Sequence in Visibility Pane

If you are having difficulty mapping via labels to actual layer names or simply wish to see a numerical order to the stackup, consider enabling the “show layer numbers” option in the color dialog. When set, the sequential layer numbers appear in the first column of the visibility pane.



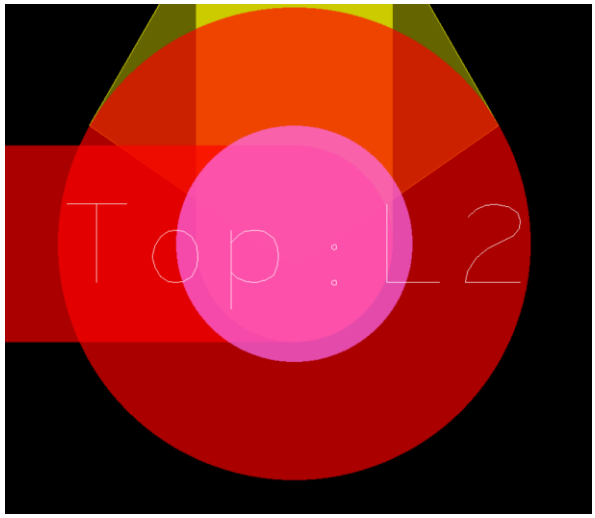


Customizing Via Labels

By default, the values associated with BB via labels begin with 1 (layer top) and increase sequentially. This may cause confusion as labels may be out of sync with the actual layer names. Consider editing the Layer ID field associated with the Cross Section Editor. There is a max 3-character limit.

Thickness <<			Physical <<					
Value mil	(+)Tol. mil	(-)Tol. mil	Layer ID	Material	Negative Artwork	No Fillet	Unused Pin Suppression	Unused Via Suppression
*	*	*	*	*	*	*	*	*
1.2	0	0	Top	Copper	☐	On		
8	0	0		Fr-4				
1.2	0	0	L2	Copper	☐		☑	☑
8	0	0		Fr-4				
1.2	0	0	L3	Copper	☐		☑	☑
8	0	0		Fr-4				
1.2	0	0	4	Copper	☐		☑	☑

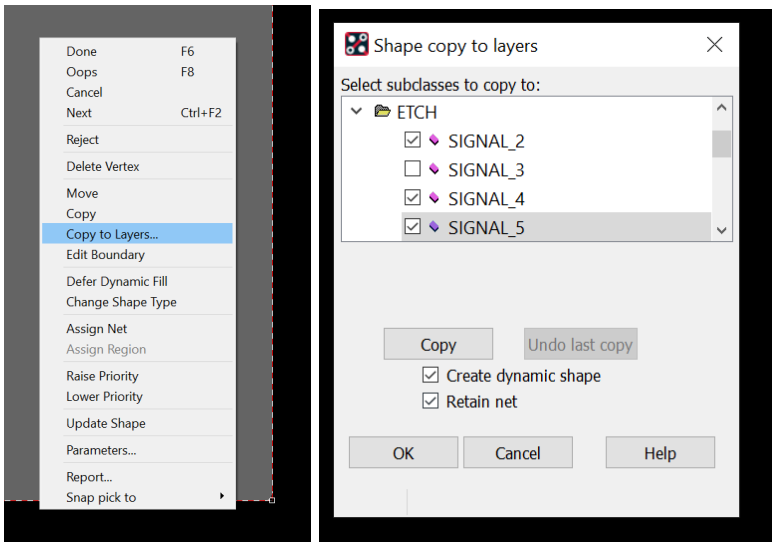
Cross Section Editor



Custom Via Label

Copy Shape to Different Layers

By now I'm sure many of you have used the Z-Copy command to copy shapes across layers. Consider a more robust option that involves first selecting a shape (Shape-Select Shape or Void) then use the RMB button to access the "Copy to Layers" command. Select the target class and subclasses then consider the options at the bottom of the form; create dynamic shape and retain net. The last step would be to click the "Copy" button, sometimes an oversite before clicking "OK"



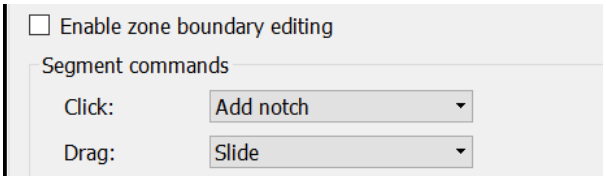
Shape Boundary Editing

If you're not already using the Shape Application Mode to edit shape boundaries, this section is a must read! The app mode is designed to improve your efficiency in the following areas:

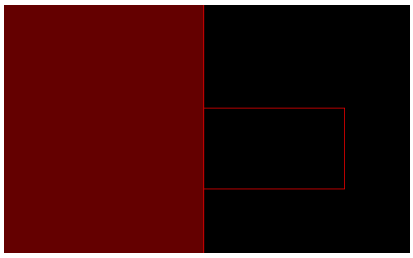
- Slide shape edges with or without corners
- Corner conversions to round or chamfer
- Multi-segment sliding
- Adding of notches

Steps:

1. Enable the app mode! Setup – Application Modes – Shape Edit
2. Find filter adjustment – recommend all elements are enabled especially shapes
3. Choose LMB click and drag behavior

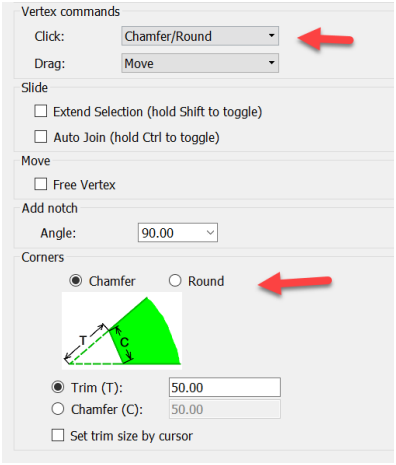


4. **Add a notch** by making 2 clicks on the shape boundary

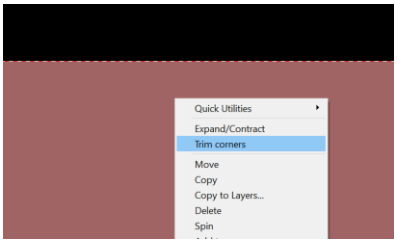


5. **Slide an edge** by holding the LMB button then drag

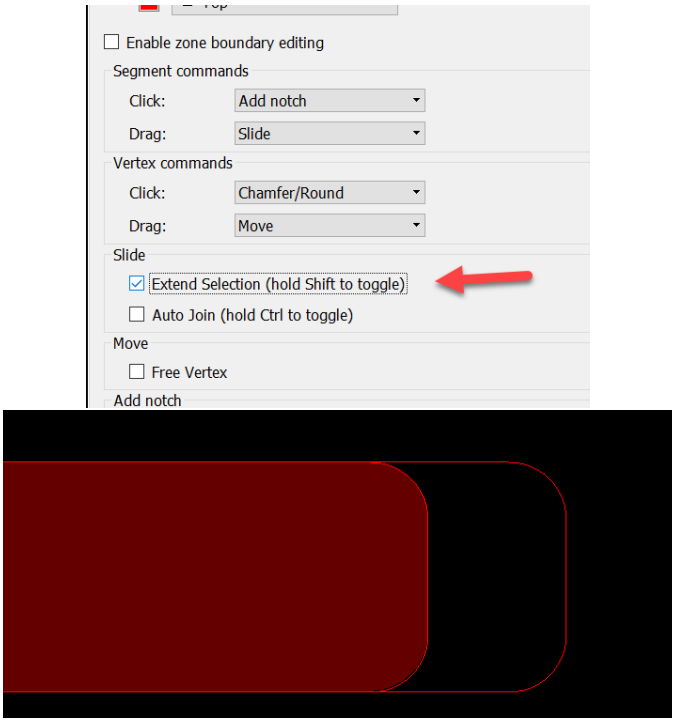
6. **Convert an orthogonal corner** by clicking the vertex location; adjust settings for chamfer or round. Note the cursor changes appearance when hovering over corner.



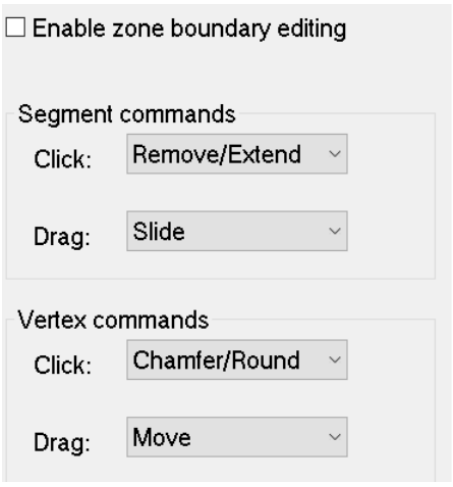
7. *Most common question – how do I **convert all corners**? Hover over shape then use RMB to access "Trim Corners" command.*

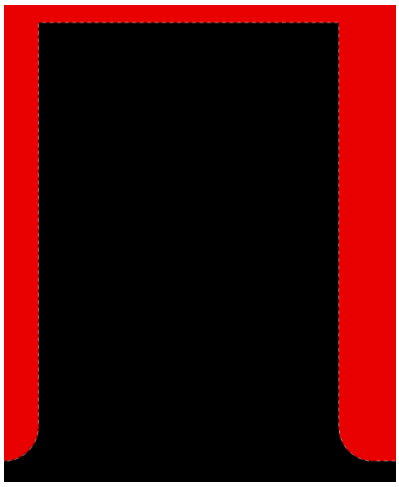


8. *Second most common question – how do I **slide an edge with round or chamfered corners**? Enable the "Extend Selection" option then use LMB drag to slide the edge with its associated corners.*



9. ***Convert chamfered/rounded corner to orthogonal*** *- adjust the "Click" option to "Remove/Extend". Select a rounded corner to revert to Orthogonal.*





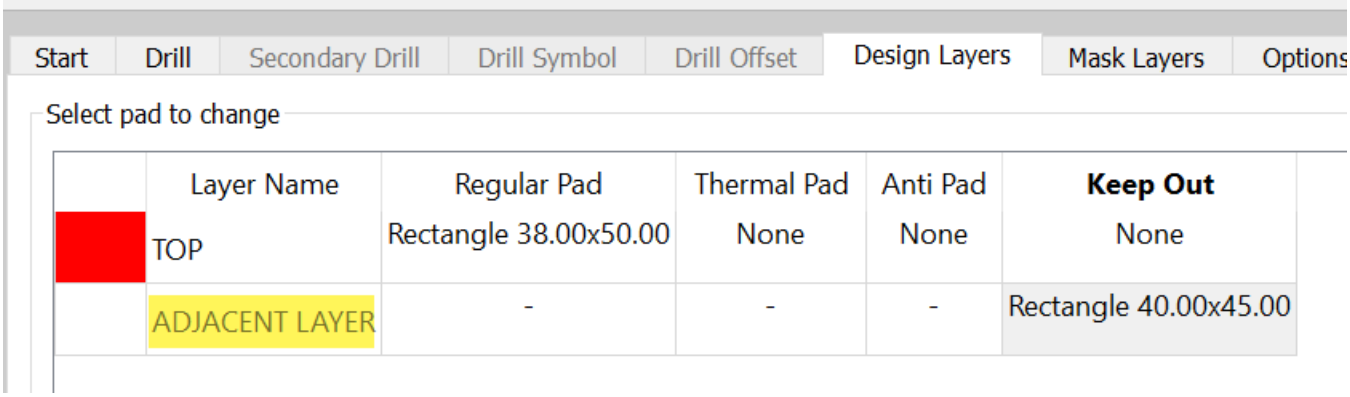
Voids under SMD Pads

The padstack supports an Adjacent Layer Keepout field that can be used for this application or even drill overshoot. Generally, the librarian controls the adjacent layer geometry in the pad definition while the Designer adds the layer depth control via the following properties.

Properties:

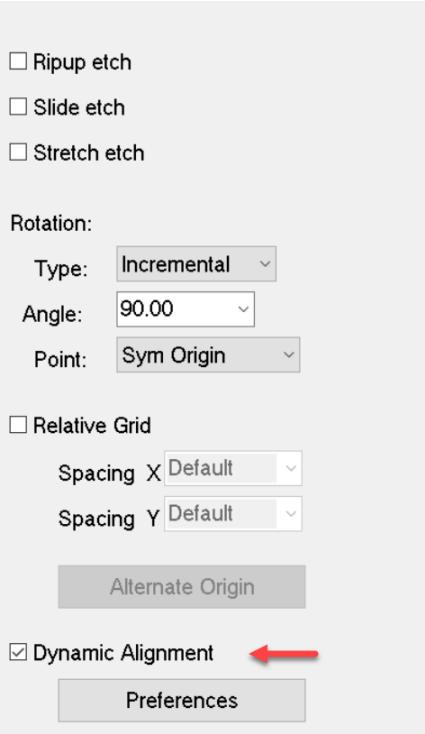
- Adjacent_layer_void_above
- Adjacent layer_void_below
- Max adjacent layer count = 8

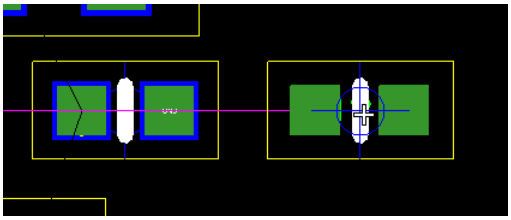
Benefit: The keepout are associated with the symbol therefore move with it!



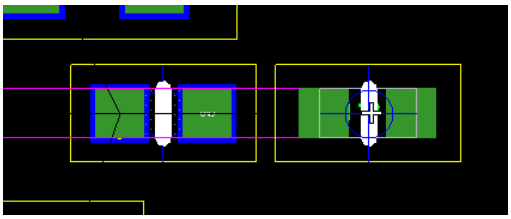
Component Placement - Dynamic Alignment

Consider enabling the Move command option “Dynamic Alignment”. When set, symbols can be aligned by its origin or placebound shape edge.





Origin Alignment

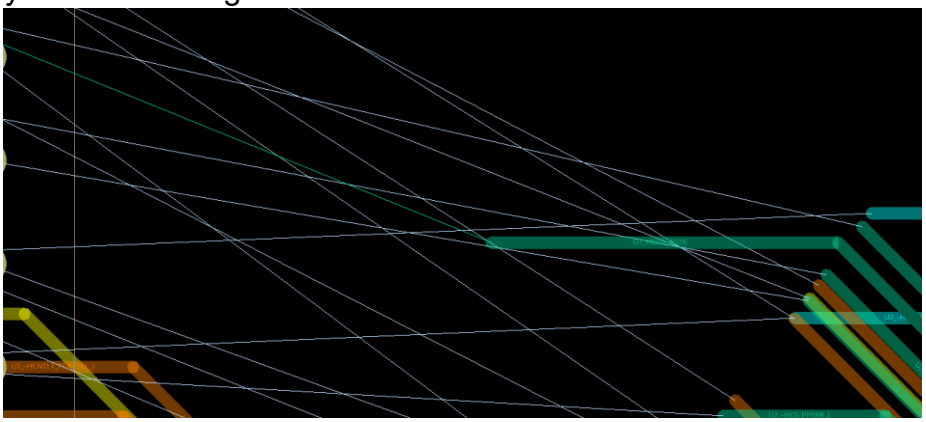


Place Bound Alignment

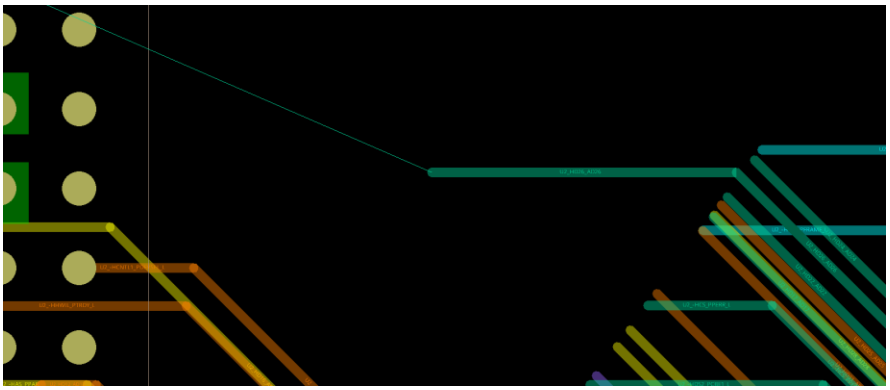
Customize the color of the alignment guides by accessing the color dialog – display folder.

Routing - Dynamic Rat Suppression

When in the Add Connect command, consider enabling the option “auto-blank other rats”. The image below shows a cluster of rats, most likely denser conditions exist on your board design.



With the auto-blank option enabled, you benefit from the temporary suppression of all rat clutter except for the active net you are routing.

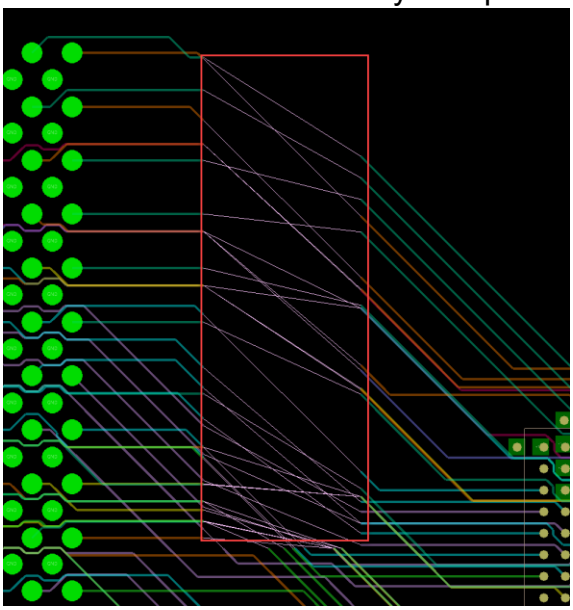


Cutting Clines

Two commands located in the Manufacture menu (OK not obvious) that could be leveraged to make cuts in your route path.

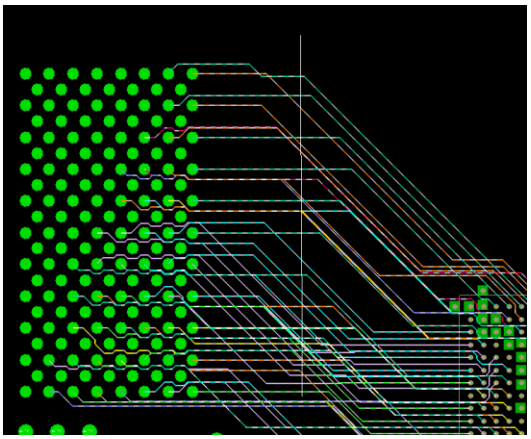
1. **Manufacture – Drafting – Delete by Rectangle**
   - Select the routes you wish to cut
   - Draw the cut rectangle

Hint – follow instructions in the command window if you experience difficultly.

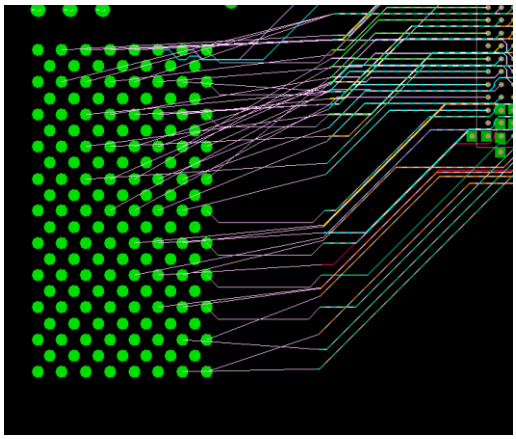


Delete by rectangle

2. **Manufacture – Drafting – Delete by Line**
   - Select the routes you wish to cut
   - Draw a line across the routes
   - Select side of line to make the cut



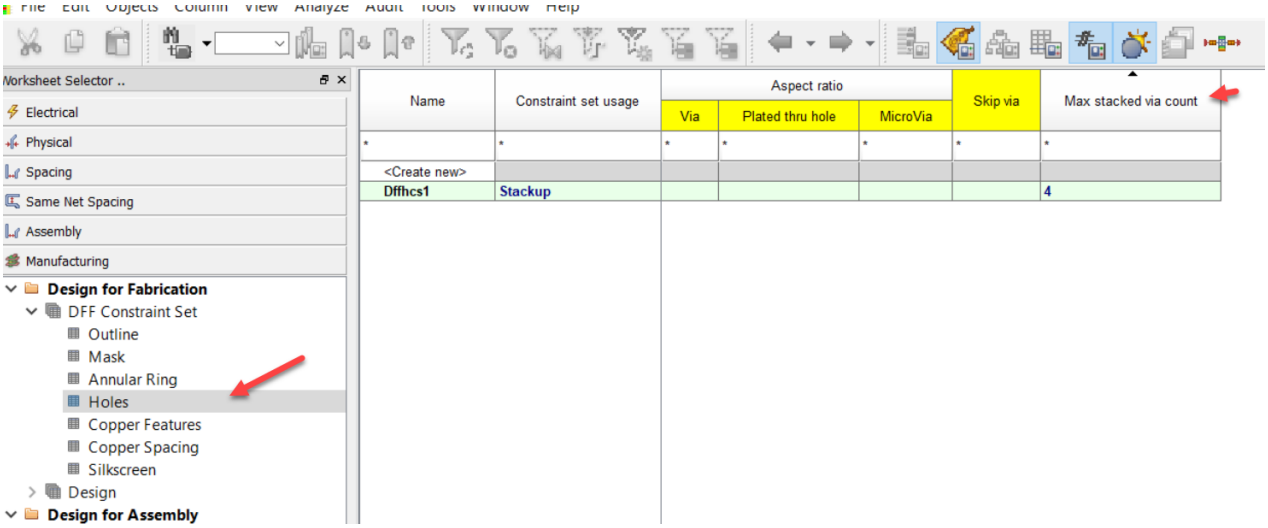
Draw line



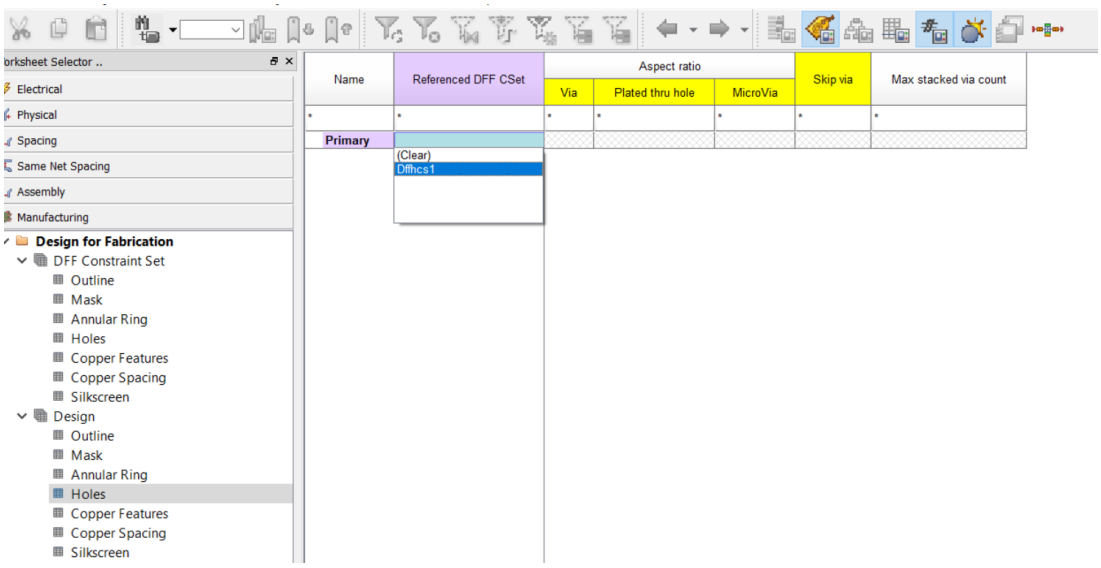
Pick side to cut

Checking for max stacked via depth

This check requires the "Venture" or "Enterprise" product where advanced DFx checks are available. Navigate to the Manufacturing domain then Design for Fabrication section. Select the category "holes" then apply the max stacked via count value to the Cset. As with other CM domains, apply this section to the Design section



The image below shows the applying of the Cset

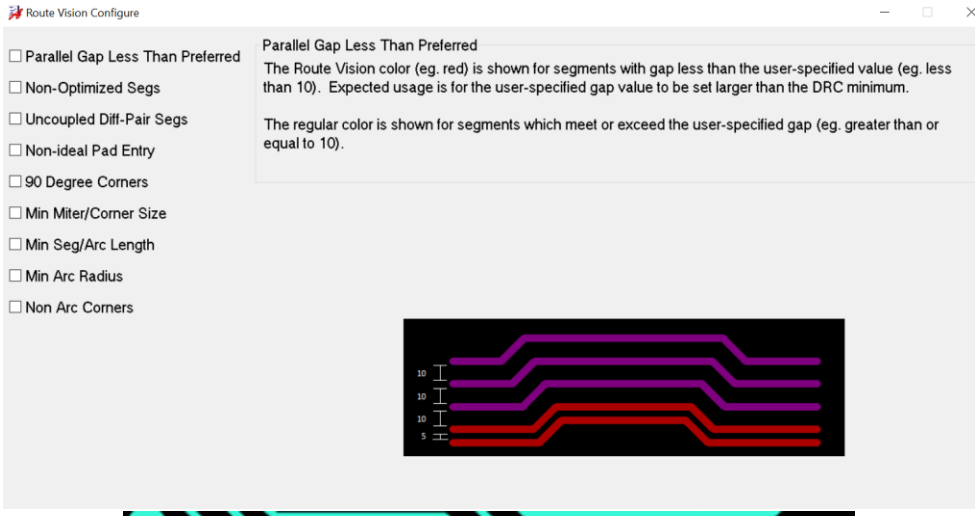


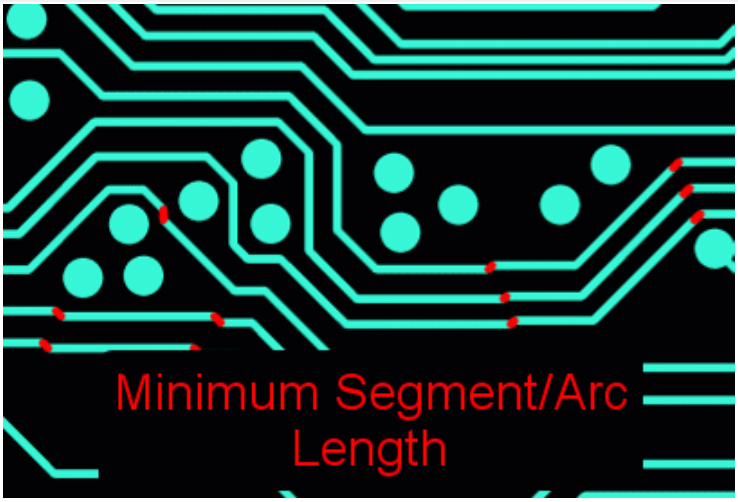
Assign net to vias

Offered previously as a skill app, general and etch edit modes now offer this command using the RMB context sensitive menu. Hover over a via(s) then RMB – Assign net to via. Refer to options pane for netlist pulldown or select net based element in the workspace.

Route Cleanup

From non-ideal pad entry, short segments, 90 degree corners or non-arc corners, consider the Route Vision Application. First enable the Vision Manager (View – Vision Manager) then in the Options Pane, select the “Visions” tab; pick Route Vision from the pulldown. Configure your settings then click OK to invoke the command.

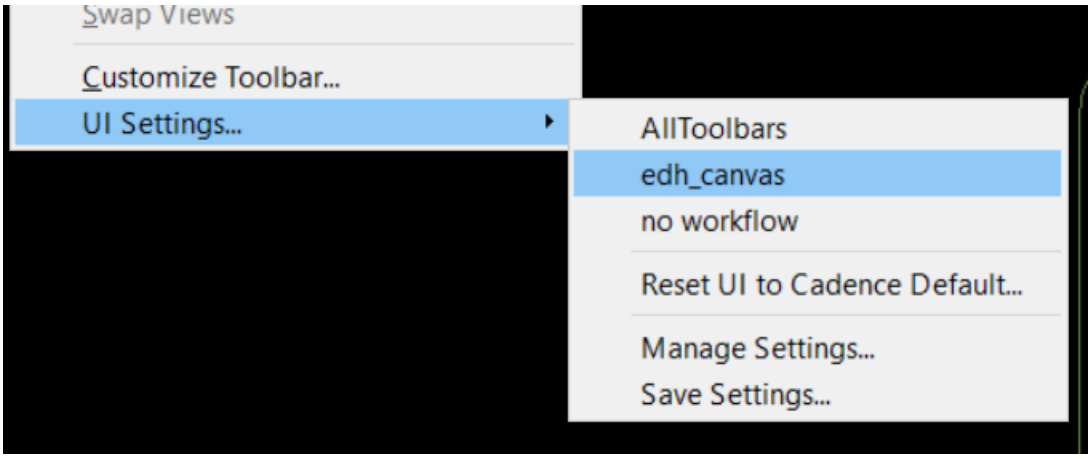




Save and Restore your Canvas settings

Allegro offers the flexible to customize the location of the option panes and toolbar icons. If for some reason your environment is reset to the Cadence default environment, you may want to consider these steps to restore your custom environment. First step would be to establish your preferred environment.

- Save Steps: View – UI Settings – Save Settings then provide name when prompted
- Restore Steps: View – UI Settings then select name from list



Reducing mouse clicks and travel

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

Center display at cursor location

funckey z "zoom center; pick -cursor"

Move your cursor to the position that you want to be the center location, and then click the z key.

Adding Vias: Use of Space Bar

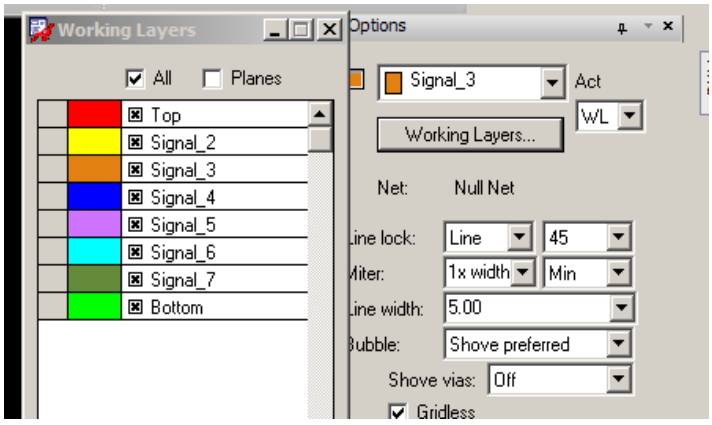
Adding a via has traditionally been done with a double-click of the LMB. Using the Space Bar can save you several thousand mouse clicks per year. The Space Bar is an easy key to click and hence best for adding vias.

funckey " " "pop bbdrill -cursor" (double quote – space – double quote)

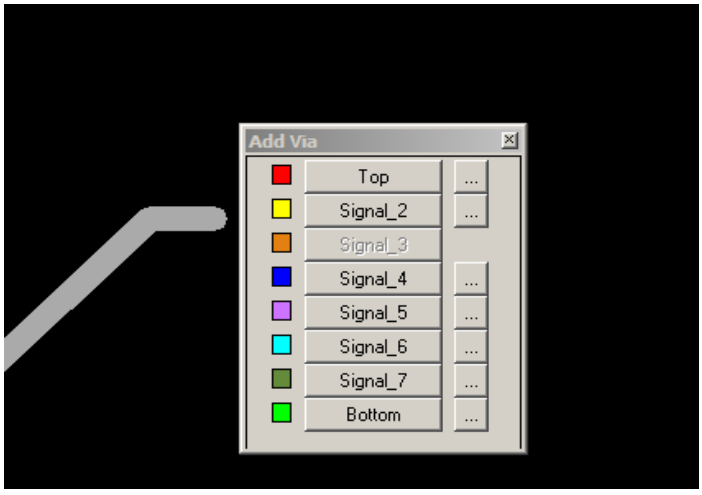
Adding Vias: “Working Layer” model

The Working Layer (WL) model is the recommended method to add conventional or HDI vias.

1. Select your target routing layer from a popup GUI that appears adjacent to the via insertion area.
2. Go into the Add Connect command, and then select “WL” over in the Options Panel.
3. A pop-up list appears. Enable the layers you want to see each time you pop a via.



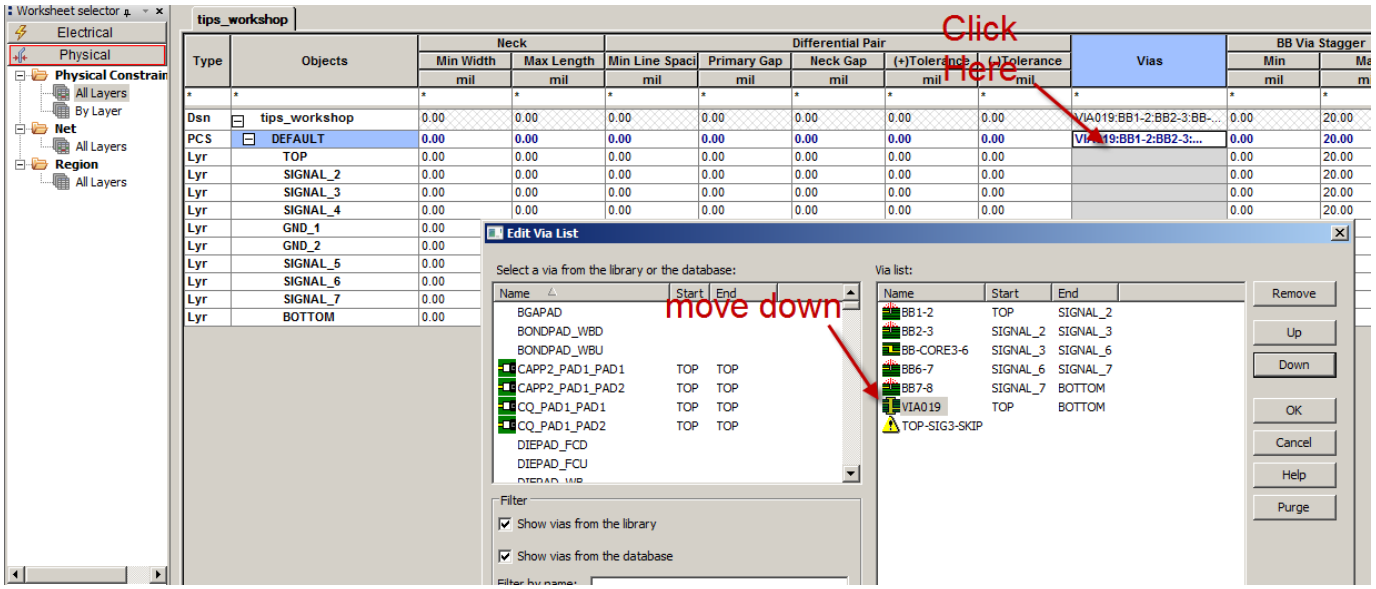
In Add Connect – Select ‘WL”



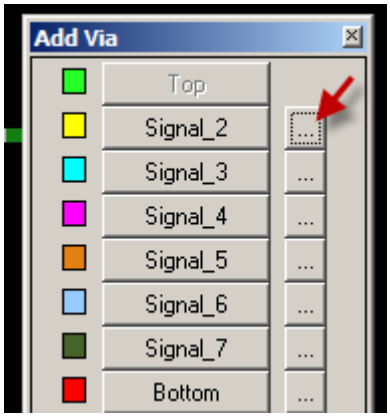
Select Target Layer during routing

What if you wanted to use HDI vias as the priority vias, and use the through-hole as an exception?

1. Open Constraint Manager and navigate to the “Physical” domain.
2. Click on the “Via List” Cell and note the order of the vias. Therefore, via019 is first used ahead of the microvias.
3. Use the down-arrow button to move via019 to the end of the list.

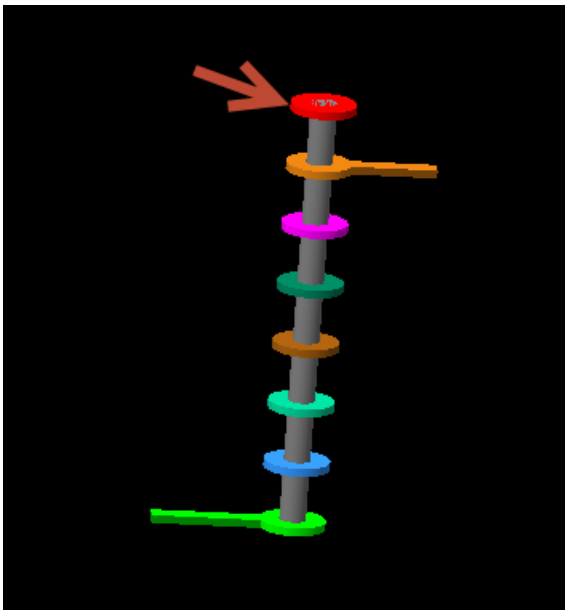


4. Add the through-hole via (via019) by selecting the (…) button. This is how we access the least preferred vias in the “via list”.



Deleting a via in a stack

1. Use Edit → Delete, select the stacked via, and then use the RMB → Split Stack command. From the popup list, select the layer pair to delete.



Add Connect: Single-click execution

In the Etch Edit Application mode, make a single pick on a pin, via or rat to begin using Add Connect. Ensure the "Enable Single Pick Execution" mode is enabled. Access this option by RMB → Customize menu.

Add Connect: Add a vertex using a Funckey key

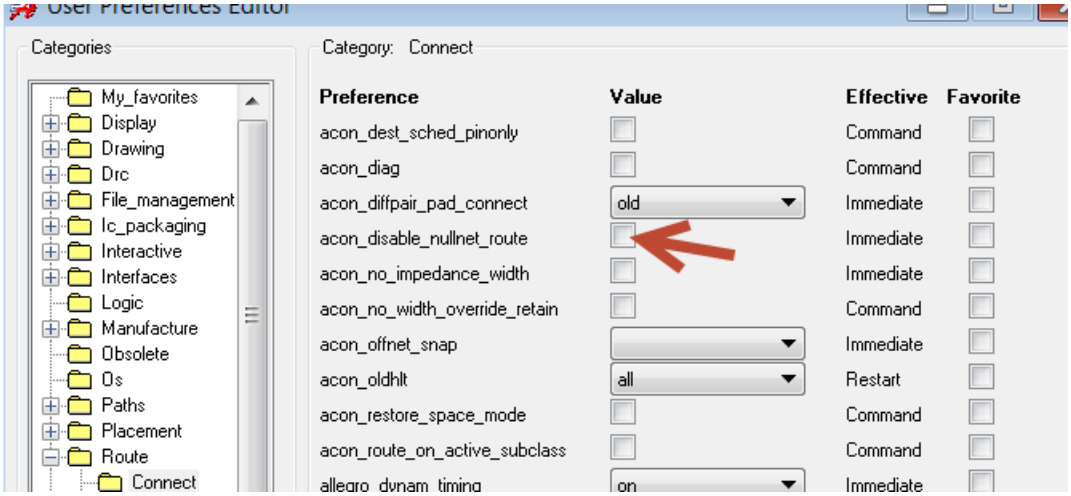
Instead of using the LMB to add a vertex point during Add Connect, consider using a Function Key. You can simply click the X key every time you want to add a vertex during routing.

```
funckey x "pick_to_grid -cursor"
```

Add Connect: Prevent null net routing

PCB Designers move the mouse a lot faster. It is easy to make a pick in open space when the intent is to pick on a pin or a dangling end segment.

1. Set the “acon_disable_nullnet_route variable in the User Preference Editor to prevent accidental picks that invoke Add Connect.



Add Connect: Toggle Corner

2. During the Add Connect command, use the F2 key to toggle the corners between arc and line.
3. Route one signal and press the F2 key four times to experience dynamic corner behavior.

```
alias a1 'FORM mini lock_mode Arc;FORM mini lock_direction 45'
alias a2 'FORM mini lock_mode Arc;FORM mini lock_direction 90'
alias a3 'FORM mini lock_mode Line;FORM mini lock_direction 45'
alias a4 'FORM mini lock_mode Line;FORM mini lock_direction 90'
alias F2 'settoggle CMD a1 a2 a3 a4;$CMD'
```

Add Connect: Add a vertex at cursor location

Instead of using the LMB to add a vertex point during Add Connect, consider using a Function Key. This will save you thousands of mouse picks per year. However, it will take some time to get used to.

You can simply click the “X” key every time you want to add a vertex during routing.

```
funckey x "pick_to_grid -cursor"
```

Consider other applications such as move components and move text that might also leverage this command.

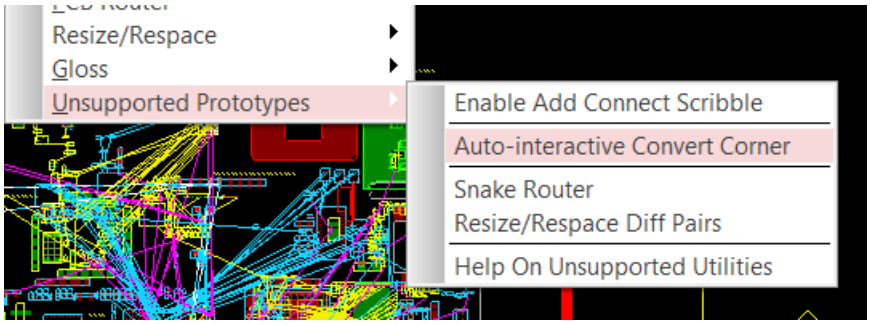
Changing line width during Add Connect

Not all the line widths are derived from Physical Csets or Regions. Consider creating function keys for the typical line widths you use that are not constraint defined. For example, the GND net may be defined as 10 mils but you might decide to widen at various locations (large pad components). This function key changes the current line width to 25 during the ADD Connect command.

```
funckey 9 form mini acon_line_width 25
```

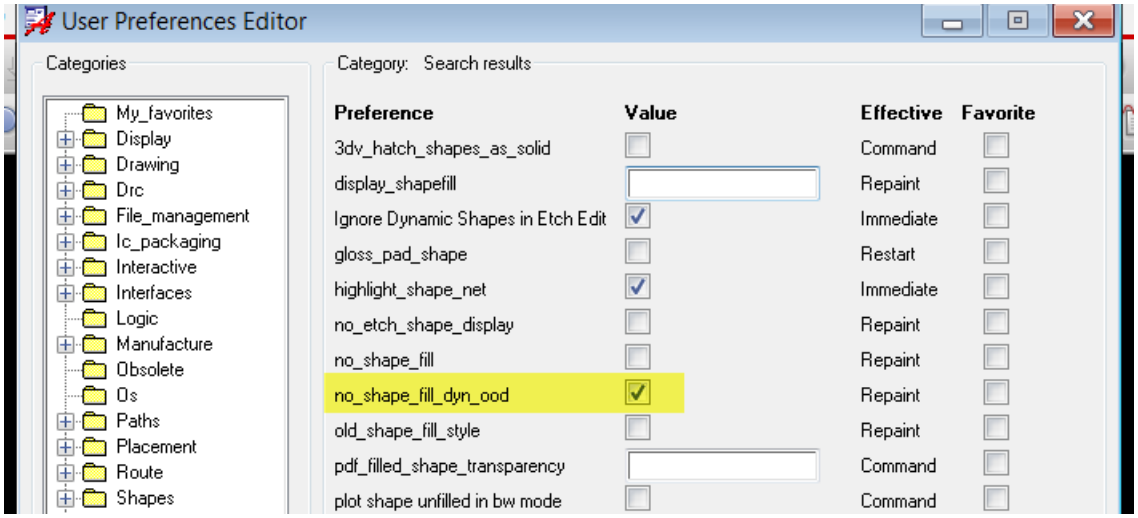
Curving the corners

1. Use Slide → Vertex Action → Arc Corner or consider the Auto-Interactive Convert Corner command if you wish to convert corners across a larger group of signals.



Disabling shape fill when shapes become disabled

This user preference variable disables the filling of dynamic shapes when these become out of date. This may reduce the need to move shapes off the edge of the board outline when setting the shape fill mode to “disabled”:



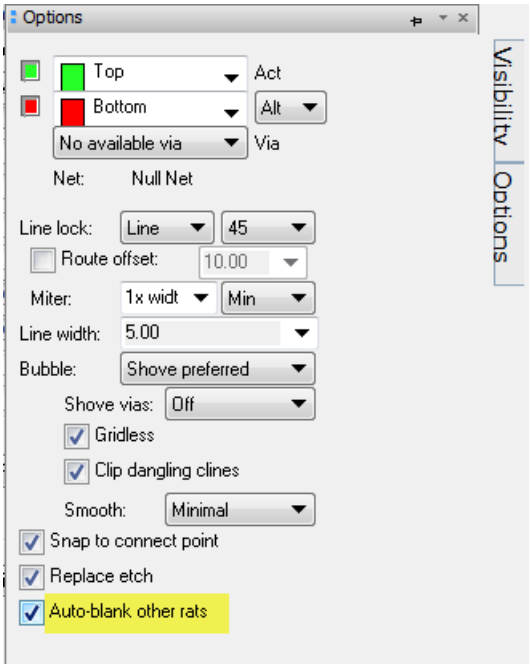
Working in Etch Edit Application Mode

1. When in “Etch Edit” Application Mode, you can:

a) Make a single pick on a pin, via or rat line to begin “Add Connect”
   i) Even move a component by holding down (drag) the LMB and press ESC key to release drag
b) Make a pick on a cline segment to invoke the “slide” command
c) Easily change the width or fix a cline or segment by hovering over it and then using the RMB context menu.

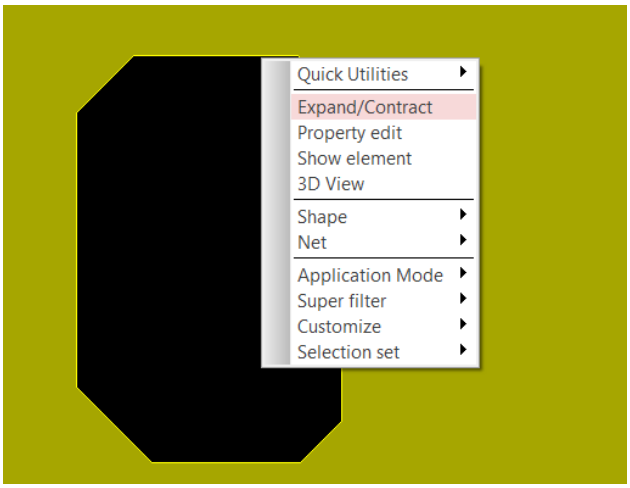
Suppress Rats during Add Connect

Want to de-clutter the screen when routing? Consider enabling the "Auto-blank other rats" option. When enabled, all rats other than the primary ones are suppressed during the Add Connect command (Introduced 16.6 QIR6).



Expand/Contract a shape or void

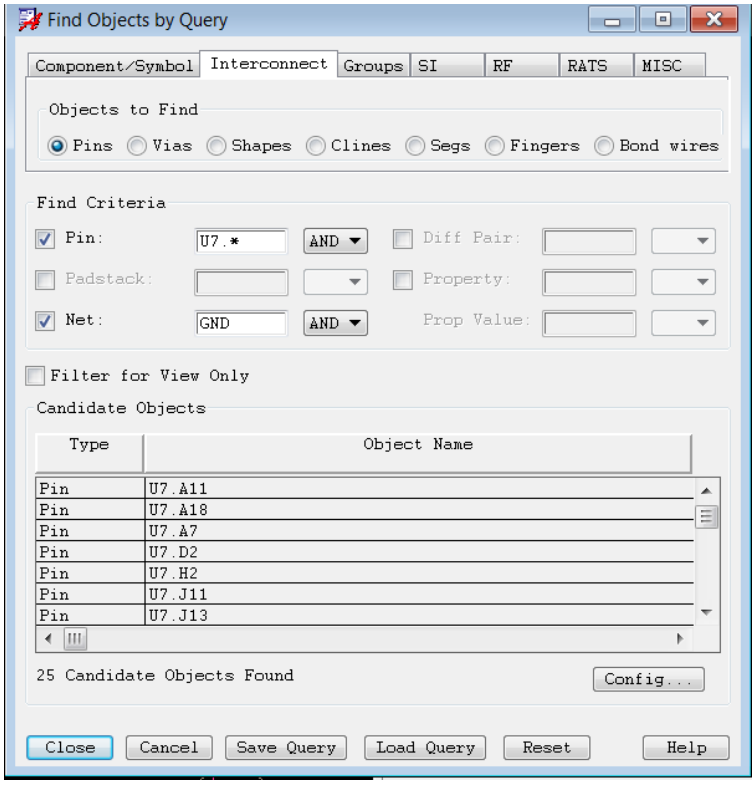
1. Select "General Edit" Application Mode.
2. Hover over the boundary of the shape/void or cavity (ensure boundary visibility is enabled in the color dialog under the Stack-Up > Conductor folder).
3. Use the RMB to access the Expand/Contract command.

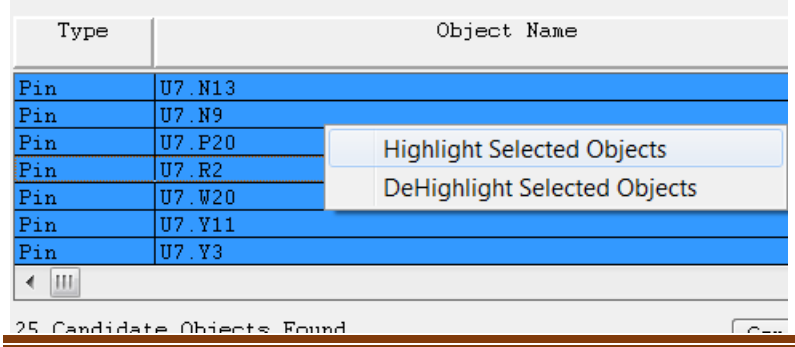


Highlight all GND pins on a particular component

1. Consider using the “Find by Query” function to locate elements based on qualifiers.
2. Invoke “Show Element”.
3. Open the Find Filter and select “Find by Query”.
4. Select the “Interconnect” tab.
5. Enable PIN and NET.
6. In the following example screen, enter U7.* in the PIN field and GND in the net field

Candidate objects are listed, and can be highlighted using the RMB to access the “highlight selected objects” command.





Slide: Single-click execution

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

Moving Components: Single-click execution

- Select Placement Application Mode.
- Make a single LMB pick on a component to move it.

Moving Component with Slide Etch option

Use the 'Move' command and enable the option "Slide etch". Works best with low pin count components like resistors.

Changing App Modes with Function Key

funckey a "settoggle CMD 'generaledit' 'placementedit' 'etchedit';$CMD"

Changing Active/Alternate Layers using Function Keys in Etch Edit Mode

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 the RMB:

RMB → Quick Utilities → Change Active Subclass

Note: If you are not in Etch Edit mode, the class must be set to “etch” for the function key to operate correctly.

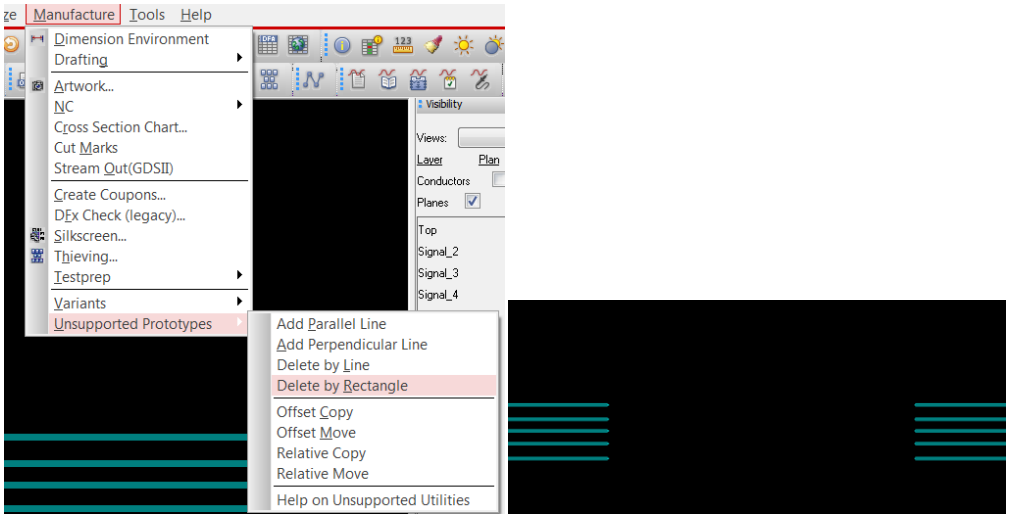
Deleting Elements using Function Key

This function key is used more than any other one. Pass your cursor over a cline, segment, via, text or shape and click “d” to delete it. No click of the mouse. Works on multiple selected elements as well.

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

Using the Delete by Rectangle command

Use the “Delete by Rectangle” command to make a window cut in a group of clines. Access the command from Manufacture → Drafting → Delete By Rectangle.



Rotating a Component using Function key

Press the function key R to rotate a component during movement:

funckey r iangle 90

Mirroring a Component Using Function key

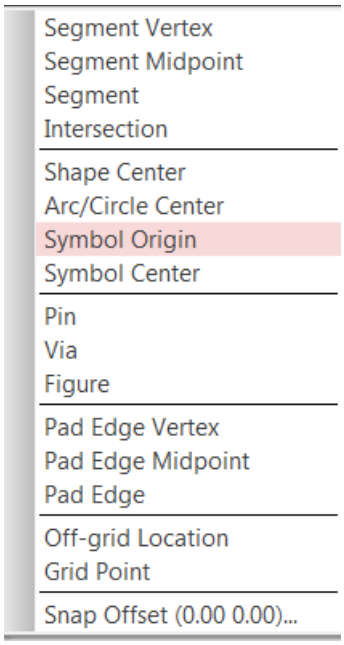
Press the function key M to mirror a component during movement:

```
funckey m "pop mirror"
```

Snapping Using Funckey key

- The “snap pick to” RMB menu contains many options. You may want to consider creating function keys for the ones you use frequently. A few examples:

```
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'"
```



Creating a “Snap Pick to” Shortcut with Scriptmode +e

- Type scriptmode +e in the command window.
- The “e” option is used to echo the Allegro command language to the command window.
- Perform a snap function and note the command is written to the ‘command window’.
- Select the command then RMB → Copy.
- Paste the command after a Funckey key and enclose in double quotes. Example:

- Funkey c “pop dyn_option_select 'Snap pick to@:@Arc/Circle Center'”

```
Pick new location for the element(s).
Command > c
CMD> prepopup
CMD> prepopup 2264.49 5210.14
CMD> pop dyn_option_select 'Snap pick to@:@Arc/Circle Center'
Snapped to (2250.00 5225.00) on 'center of arc seg: xy (2225.00 5225.00) xy (2225.00 5225.00)' using 'Arc/Circle Center' mode.
last pick:  2250.00 5225.00
CMD> trapsize 1834
CMD> placementedit
CMD> prepopup 2264.49 5210.14
CMD> pick grid 1571.30 5580.58
last pick:  1571.00 5581.00
No element found.
CMD> zoom in 1
```

Mirroring a stationary component (do not move it)

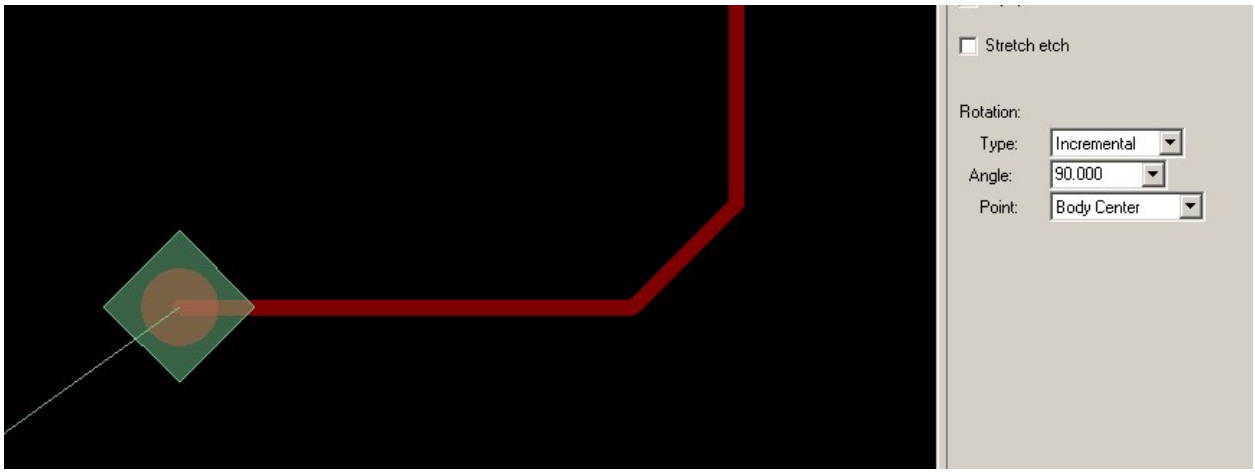
While hovering over a symbol, use the RMB and select the Mirror command.

Aligning Components

1. Window-select a group of components, hover over the “anchor” component, and then use the RMB to access the “Align components” command.
2. The key point with hovering is you must be hovering over an object. Do not hover in BLACK SPACE!
3. Consider the options for equal spacing or compress to DFA spacing.

Snapping a rat T to a pin/via

While moving a Rat T, try setting rotation point to “Body Center”:



Protecting Alias/Function keys

Is there is a way to prevent users from creating aliases that are named the same as Cadence aliases, or the ones originated from site admins?

- To protect (for example, read-only) alias/funckey assignments, use the alias_protect command.

Usage:
alias_protect <alias/funckey>

Example:
- funckey m move
- alias_protect m

  Commands assigned the Alias_Protect option will always win out over similar commands sourced from other locations.

Extracting single-footprint from PCB Database

1. Use the skill command to write out the package symbol data files for a single symbol. In the following example, SR0201.dra and SR0201.psm are generated along with the respective padstacks.
2. Enter the following command in the Allegro command window:

   skill axlWritePackageFile(car(axlSelectByName("SYMTYPE" "SR0201"))->definition)

Refreshing package symbol instance

1. Select Placement Edit App Mode.
2. Hover over package symbol, and then use the RMB to access the “Refresh Symbol Instance” command.

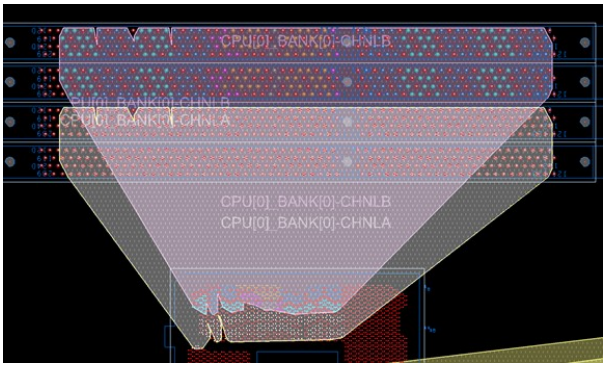
Using Mouse Wheel to control interface visibility

Net-Groups are part of the Base Allegro/SIP product line, and provide the user with graphical and CM “Objects” that are used to “map” electrical interfaces to physical objects the layout designer can easily understand. This mapping provides convenient grouping of nets that have similar electric, physical, and spacing route path requirements.

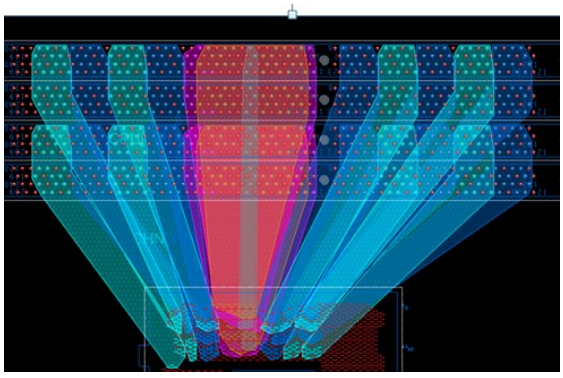
The following two “button” commands let you use the middle mouse wheel while holding the Shift key to ascend or descend the net-group hierarchy:

button Swheel_up interface_vis up
button Swheel_down interface_vis down

Objects			Referenced Electrical CSet	Verify Sched
Type	S	Name		
*		*	*	*
Dsn		Netgroup_Demo_Start		
NGrp		CPU[0]_BANK[0] (2)		
NGrp		CPU[0]_BANK[0]-CHNLA (4)		
NGrp		CPU[0]_BANK[0]-CHNLB (4)		
NGrp		CPU[0]_BANK[0]-CHNLB_ADDR_CMD (24)		
NGrp		CPU[0]_BANK[0]-CHNLB_CLOCKS (4)		
NGrp		CPU[0]_BANK[0]-CHNLB_CTRL (16)		
NGrp		CPU[0]_BANK[0]-CHNLB_DATA (9)		
NGrp		CPU[0]_BANK[0]-CHNLB_DATA-BYTE[0] (10)		
NGrp		CPU[0]_BANK[0]-CHNLB_DATA-BYTE[1] (10)		
NGrp		CPU[0]_BANK[0]-CHNLB_DATA-BYTE[2] (10)		
NGrp		CPU[0]_BANK[0]-CHNLB_DATA-BYTE[3] (10)		
NGrp		CPU[0]_BANK[0]-CHNLB_DATA-BYTE[4] (10)		
NGrp		CPU[0]_BANK[0]-CHNLB_DATA-BYTE[5] (10)		
NGrp		CPU[0]_BANK[0]-CHNLB_DATA-BYTE[6] (10)		
NGrp		CPU[0]_BANK[0]-CHNLB_DATA-BYTE[7] (10)		
NGrp		CPU[0]_BANK[0]-CHNLB_DATA-ECC (10)		



Top level View



Byte Lane Level

Aliasing commands to the Middle Mouse Wheel

The “button” command can be used to alias the Middle Mouse Wheel to commands. Works with SHIFT, CONTROL and SHIFT + CONTROL combinations.

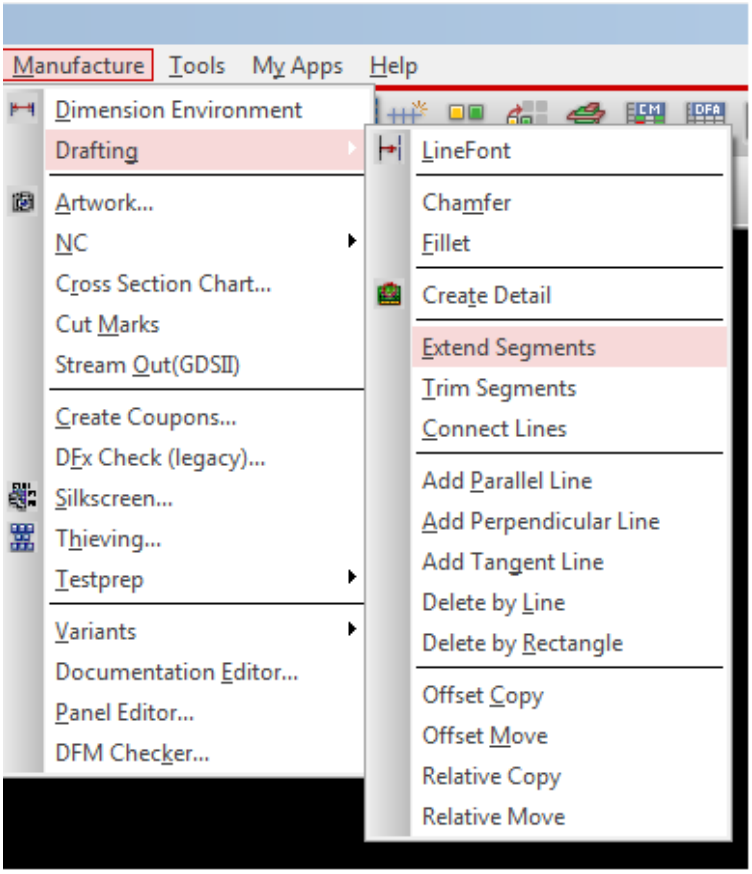
Examples:

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, the experienced designers can use the <Control> Middle Mouse Wheel to increase the font size.

Trimming and Extending Drafting Lines

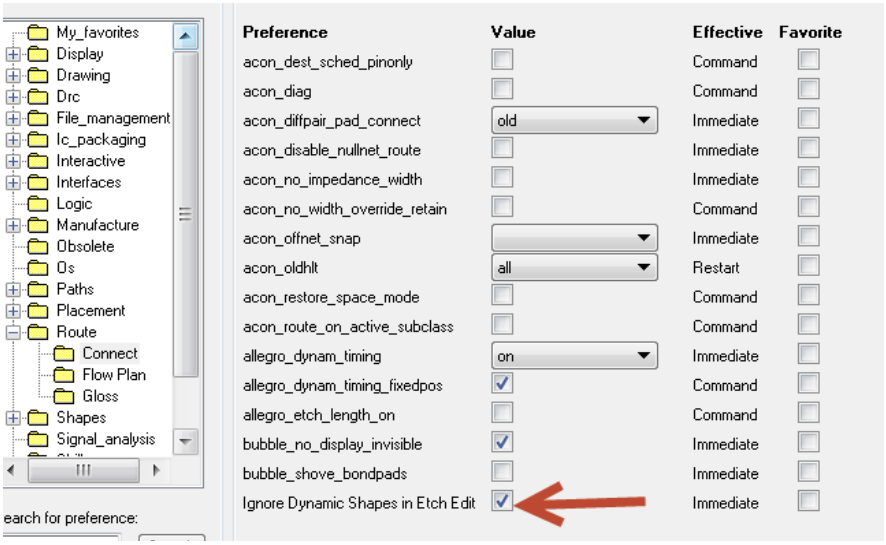
Access these commands from Manufacturing → Drafting Menu (This is available from one of the QIR's for SPB 16.6).



Sliding vias/clines freely into dynamic shape

This variable when set will let you freely slide vias and clines into dynamic shapes. Without the variable, you will experience "jumpy" behavior at the boundary. Set the "Ignore Dynamic Shapes in Etch Edit" variable.

This has been made a default behavior in 16.6. The variable is now in the User Preference editor.



Variations of a Route Keepout Area

- There are a suite of properties that permit exceptions for route keepout areas
    - Allow vias - apply the property vias_allowed to KO shape
    - Allow Shapes – apply the property shapes_allowed to KO shape
    - Allow Routes – apply the property routes_allowed to KO shape

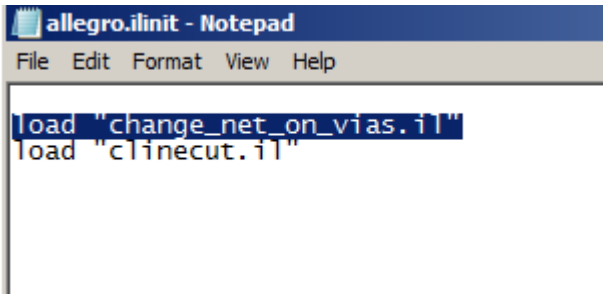
Replacing padstacks by window selection

In General Edit Application Mode, window around the set of pins or vias you wish to change, and then use the RMB → Symbol Pin or Via → Replace Padstack → Selected Instances command.

Changing Net Names on vias (skill code)

A skill application named “change_net_on_vias.il” is located in your install directory share/pcb/examples/skill/cmds.

1. If not already done, open your allegro.ilinit file (located in your PCBENV dir).
2. Add this command and then load “change_net_on_vias.il”.



3. Enter change_vianet in the command window to launch the application.

Polygon selection window

If a polygon selection window is desired:

RMB → Selection Set → Select by Polygon

Z-Copy: A powerful utility

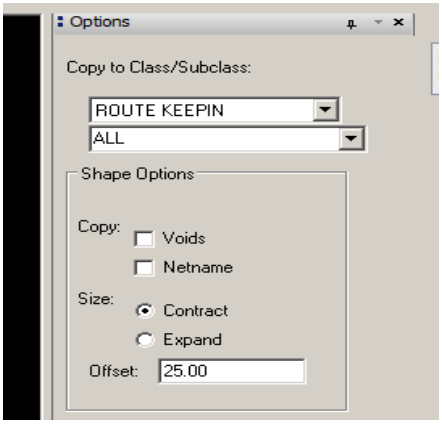
Creating a Route Keepin area derived from the Board Outline

Create a Route Keepin area derived from the board outline. Most of you might know how to do this. There is a handy tip following this short step.

1. Select Edit Z-COPY, and then adjust the options settings as follows:

   - Class = ROUTE KEEPIN
   - Subclass = ALL.
   - Enable 'contract'
   - Enter value of 25 in the Offset field

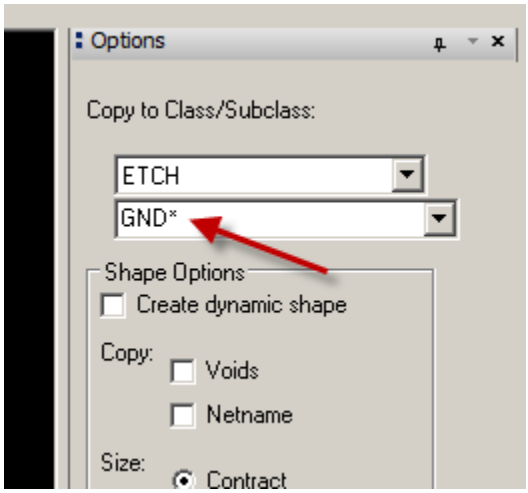
2. Select the Board Outline to create the Keepin shape.



Tip: Z-Copy can be used to target multiple layers in one operation?

If, for example, you wish to create multiple GND planes, the names of the layers may be called GND_1, GND_3, GND_5, GND_8, and so on.

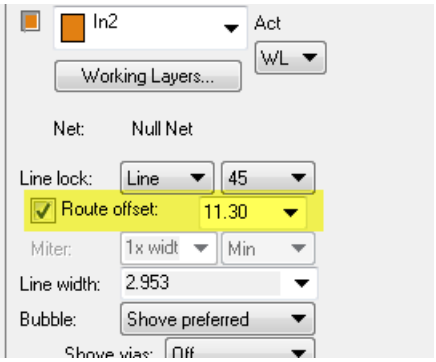
3. Select Edit -Z-Copy.
4. Adjust the options form so that Class = Etch.
5. Select one of the GND layers.
6. Edit/backspace the numerical part of the layer, and then enter the wildcard '*' character as shown in the following figure.
7. Set Offset value to 0.
8. Select the Route Keepin to create the 2 GND planes.



Enabling offset routing (fabric weave)

For routing that requires offset angles (11.3 typical), while in the Add Connect command, click the W key to enable the “Route Offset” option.

funckey w "pop flip"



Fixing/Unfixing elements in the design

Tips to add and remove the Fixed Property.

Quickly unfixing all elements

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

1. Click the Unfix Icon.

2. RMB → Unfix All.

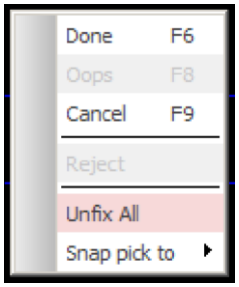


Figure 5

Fixing the location of a symbol

Mouse over the symbol, and then select the “Fix” command from the context sensitive RMB menu.

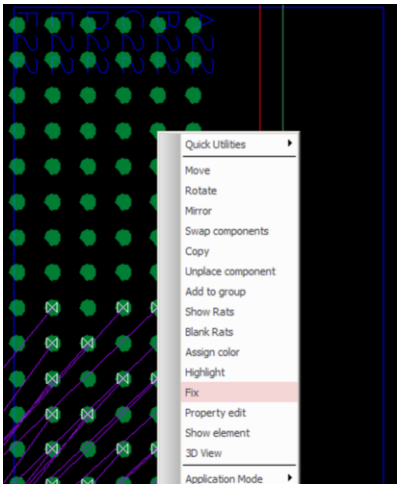


Figure 6

Datatip Display of the Fixed Property

By hovering over it, you can know whether an element is fixed.

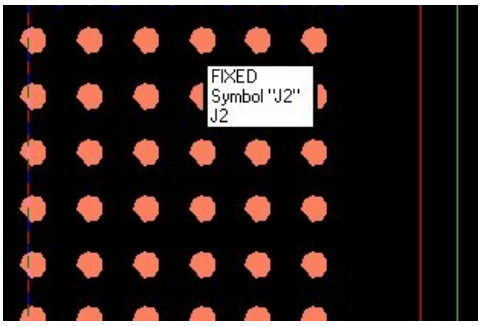


Figure 7

Configure the datatip setting as shown in the following figure. You might also want to enable other symbol-related properties to display in the datatip window. This is done from the Setup → Datatip Customization menu. Select Symbol Instance under Object Type, and then enable the value “FIXED”.

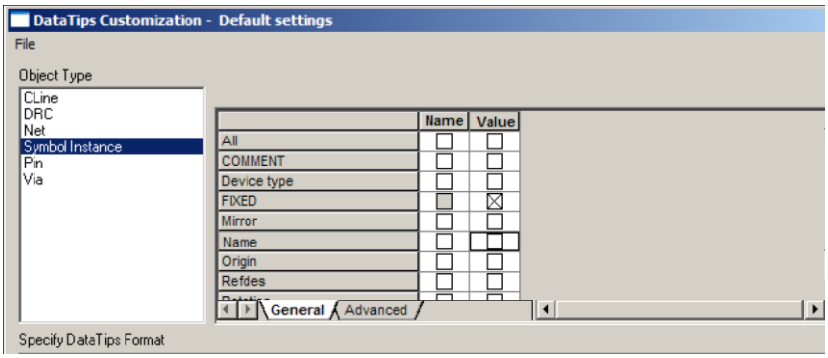


Figure 8

Datatip Options

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, when you type in the word “datatip” and click Search, variables to delay, locate and disable datatips are available to you.

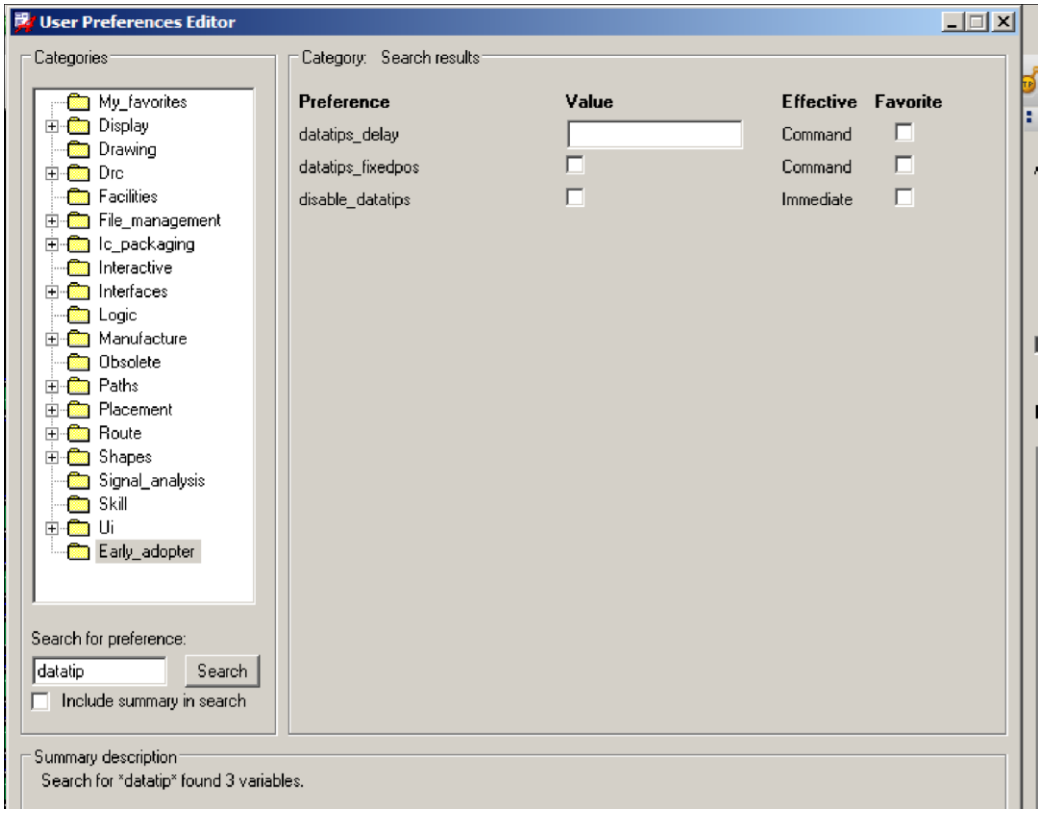


Figure 9

The complete set of variables related to Datatips is located in the Display → Datatips Category.

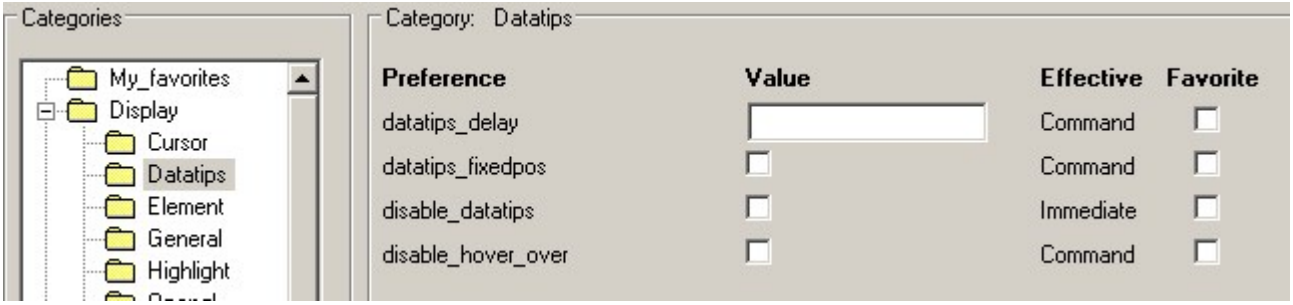


Figure 10

status Form Traffic Lights

You can click on the indicators to obtain a report?

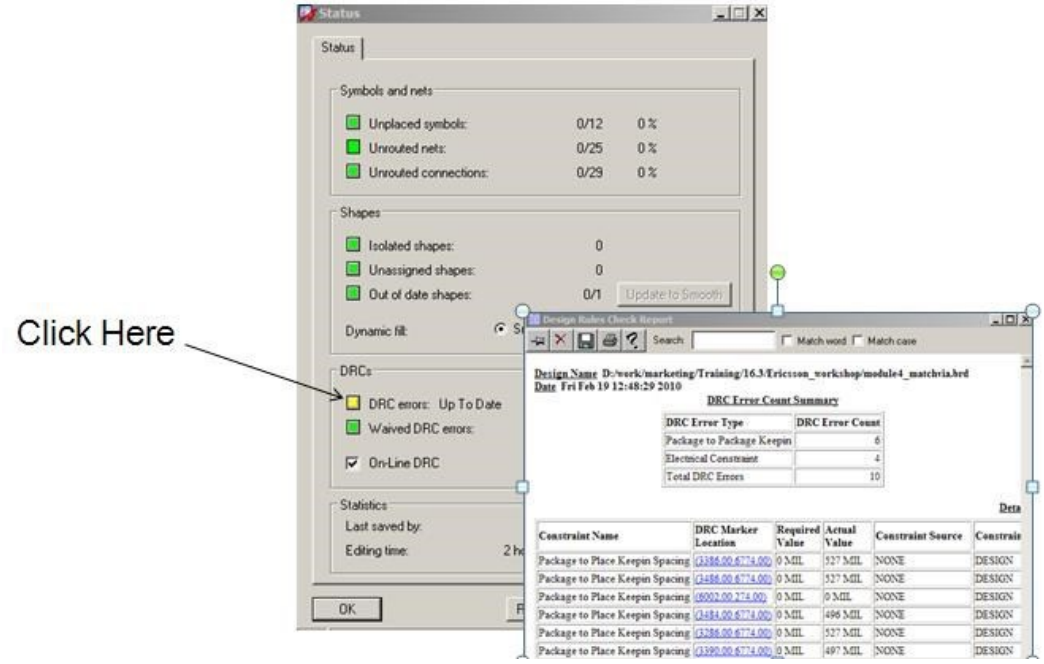


Figure 11

"Go to" X,Y Location

Navigate to an X,Y location in the Design

You wish to quickly go to a specific X, Y location in the Design. For example, you want go to the location x = 5000, y = 7000.

Adjust your zoom level, and then click on the "P" button located at the base of the canvas display. Enter in your coordinates then click "Pick".

Note: Set application mode as none for this behavior.

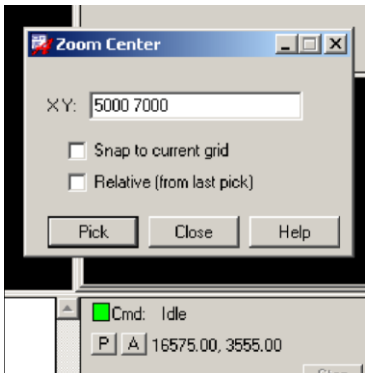


Figure 12

Disabling Custom Color Assignments

You may inherit a board from another designer or shop and question some of the color assignments. One quick method to disable color overrides that might have been applied to nets, buses, Diff Pairs is to go into the color form. If you select the "Nets" radio button on top, you will see a new button called "Disable Custom Colors". Select this option to revert to "layer-based" coloring.

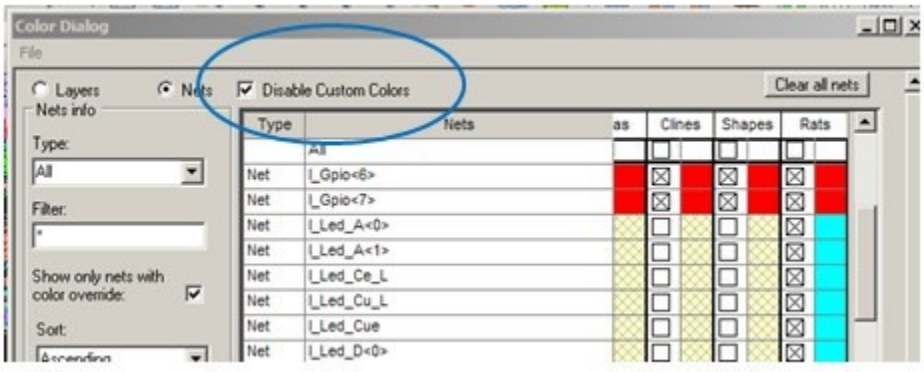


Figure 13

Color Dialog: Open the last used folder

Set the “color_lastgroup” variable located in Setup → User Preferences → UI → Control_panel.

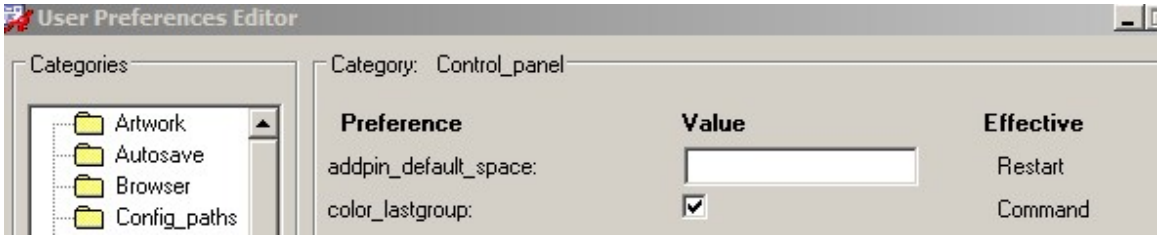


Figure 14

Displaying a layer

Click the color swatch box to the left of the subclass to quickly turn layers on/off during a command.

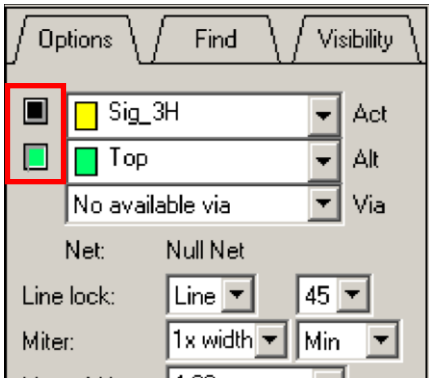


Figure 15

Invoking Allegro in No OpenGL mode

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

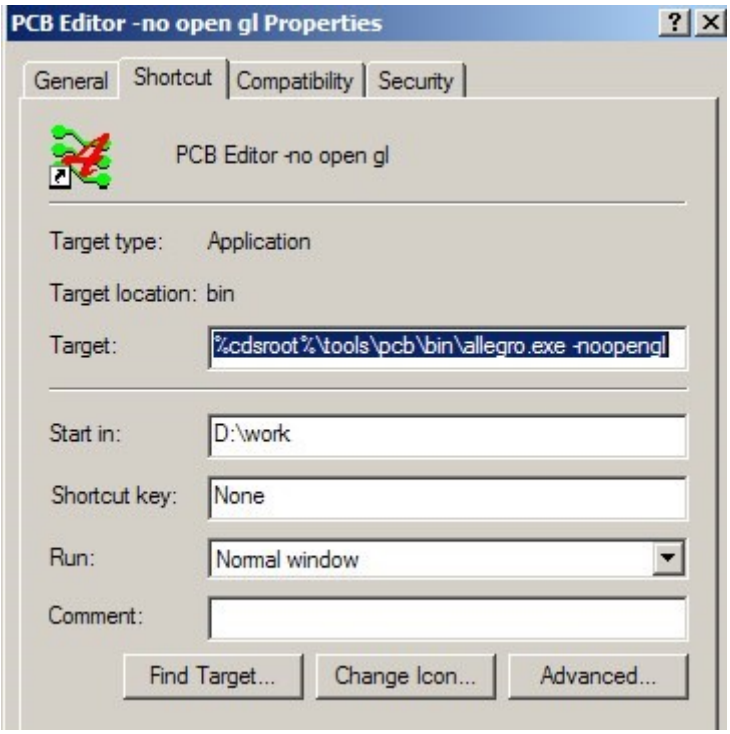


Figure 16

Displaying properties in the canvas

Any property attached to an element can be displayed in the canvas. From the Display → Property menu, select the Graphics tab, and then select properties to display from the left column. Clicking the Create button writes out the property values and/or the respective name to the subclass properties.

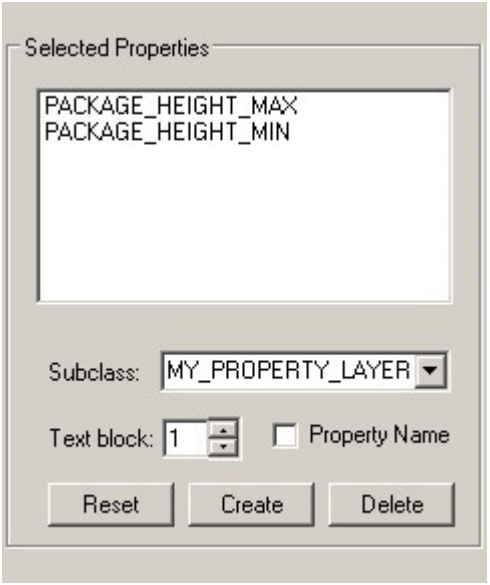


Figure 17

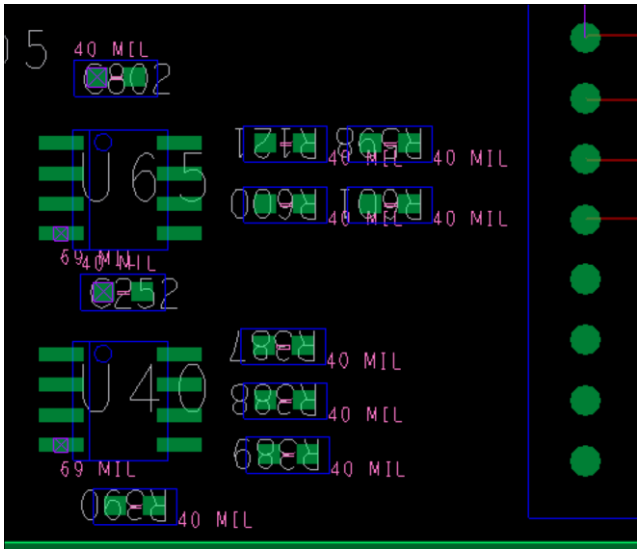


Figure 18

Suppressing pin-to-pin violations within the same symbol

PCB Designers often use Constraint Regions to suppress pin-to-pin violations within the same symbol. The “nodrc_same_sym_pin” property can be applied to the symbol. However, 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. Go to Edit → Property and then select drawing in the Find by Name field.

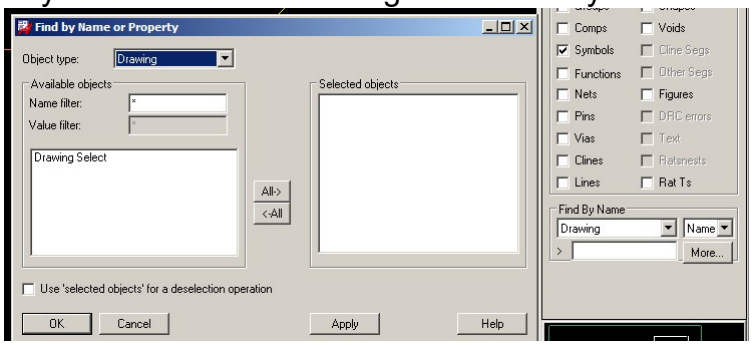


Figure 19

2. Select Drawing Select. I has now moved to the selected objects column.

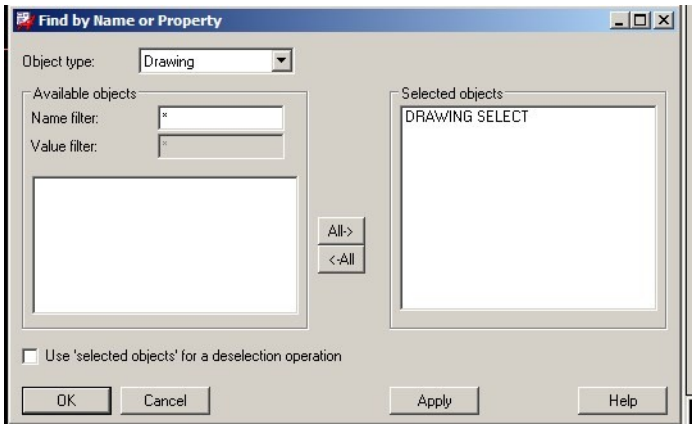


Figure 20

3. Click Apply and then pick nodrc_same_sym_pin from the list.

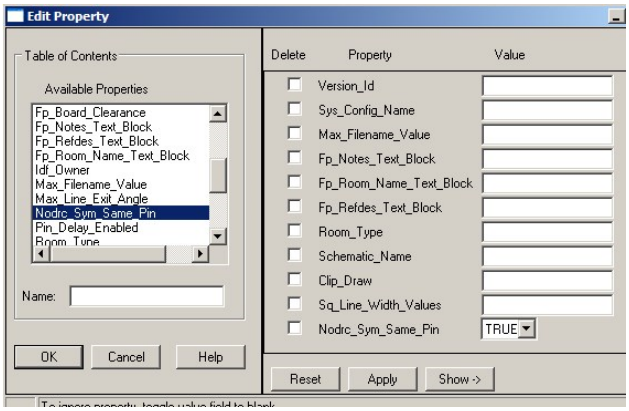


Figure 21

Creating a Command Shortcut with Scriptmode +e

Adding a drawing level property in the previous tip was labor intensive. Now you can create a shortcut.

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

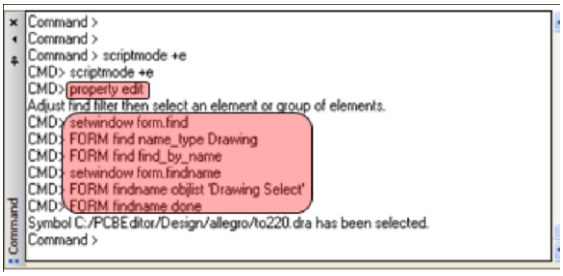


Figure 22

2. Copy all the commands above to a single line.
3. Separate with semicolons.
4. Map to a function key. When using multiple commands in a function key or alias, put these in double quotes.
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, and then press “ENTER” in the command window to launch the property editor dialog.

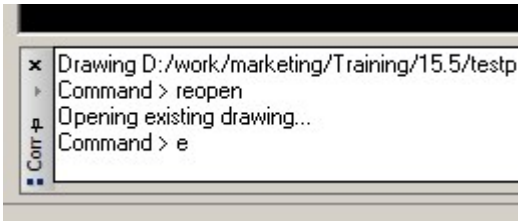


Figure 23

Scriptmode +i

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

Suppose you want to create an alias to toggle end cap display but do not want to see the parameter form appear while executing the script, use:

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

DFA Dynamics: Controlling spacing bubble resistance

When placing components to DFA rules, it is difficult to pull back the component when the bubble appears.

The resistance can be controlled by the “dfa_pause_level” variable located in User Preferences → DFA_DRC. Set it to 0 for no resistance.

“Locked” Property

The Locked property was designed for Re-Use modules. However,:

- 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 or deletion of the assembly outline or other data elements.

Suggestion: Consider applying the locked property to the symbol definition (at the library level).

Locating a dynamic shape

A dynamic shape is "out of date" BUT has a "No Etch" status.

You cannot find the shape in question. It says it is at a particular XY location, on a particular layer, but there is nothing there. Turn on the “Boundary” color item for that layer to locate the shape outline.

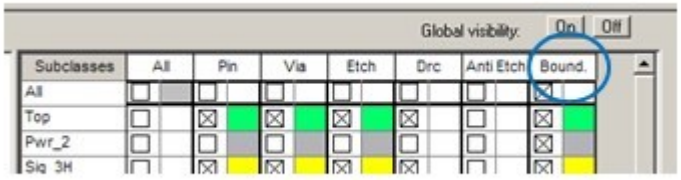


Figure 24

Voiding of GND Clines through GND Plane

Here is the method to auto-void GND sense lines being routed through a GND plane. In the following example, a GND net is routed through a GND shape. The 'void_same_net' property was applied to the cline.

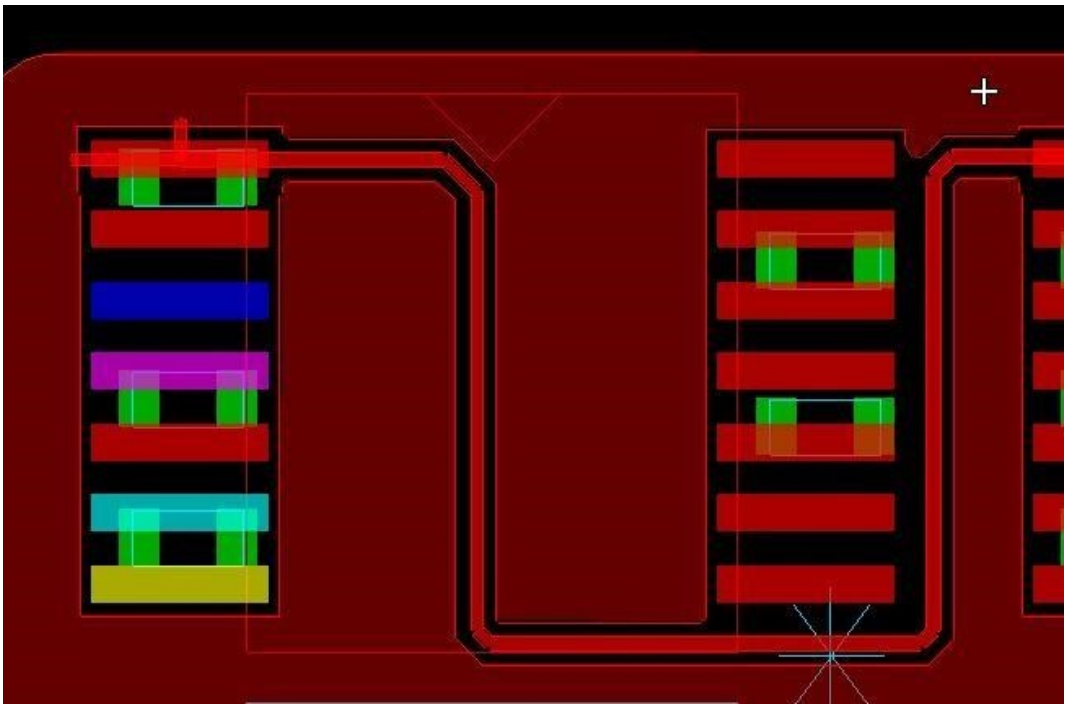


Figure 25

How to convert non-Acute angles for shapes to round corners?

This feature is available in 16.3. Go to the dynamic shapes parameter and set the Acute angle control to "Full Round". Static shapes get the same capability in a future ISR with the eap_static_newsmooth env variable.

HTML Reports

Reports and messages can be displayed in HTLM format when the "allegro_html" variable is set. One advantage of using HTML is the ability to search through the report.

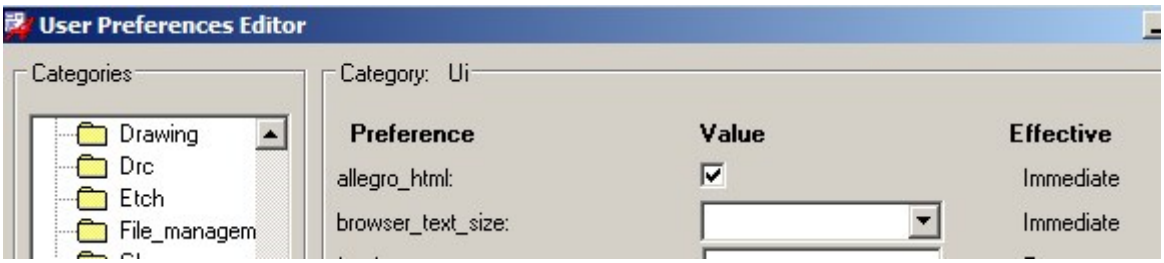


Figure 26

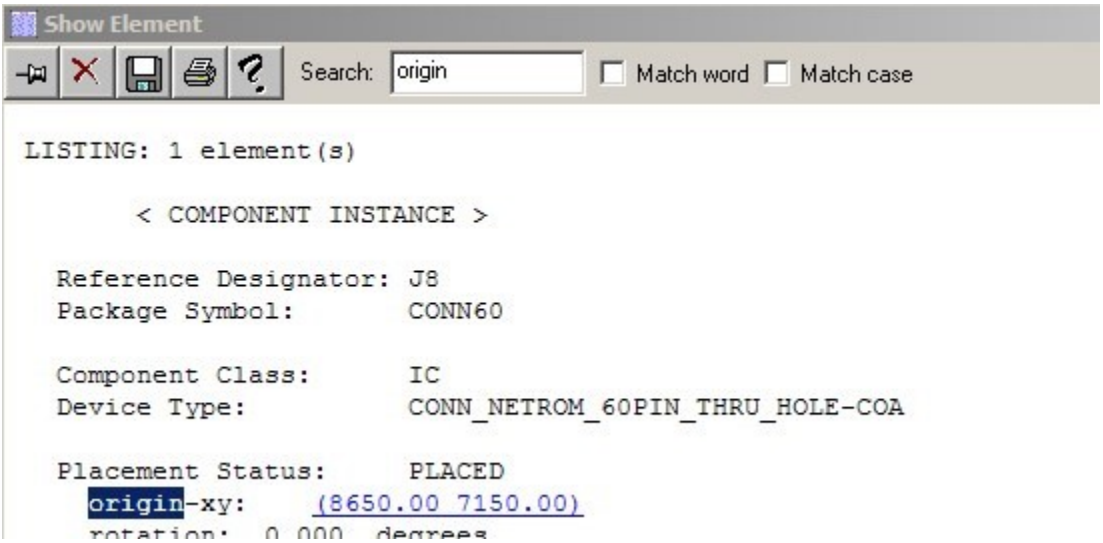


Figure 27

Printing a form using Control+P keys

Did you know Control+P can be used to print a form?

For example, inside the DFA Symbol classification window, press Cntrl+P to open up the Print form window, and then click the "To File" button to save the information to a text file.

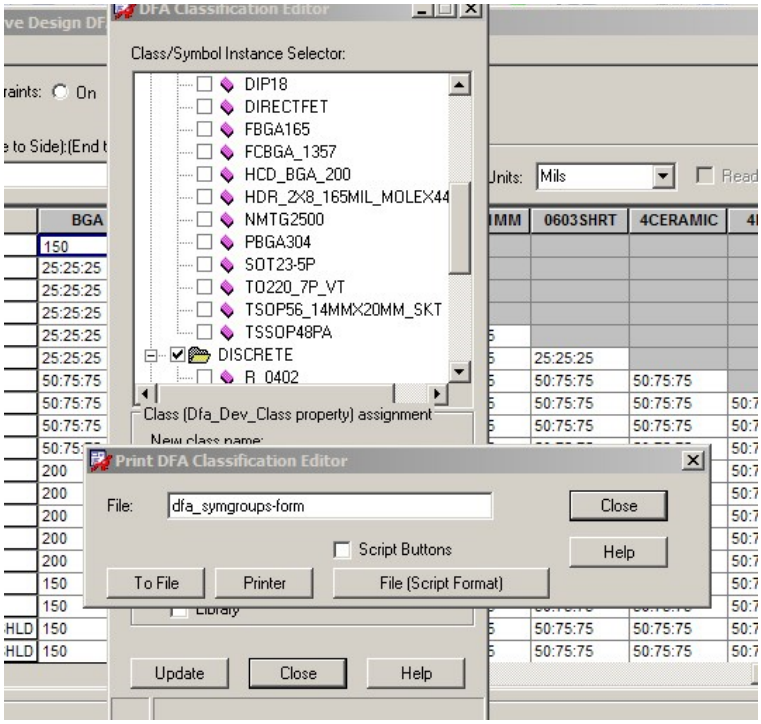


Figure 28 - Control P the click "to file"

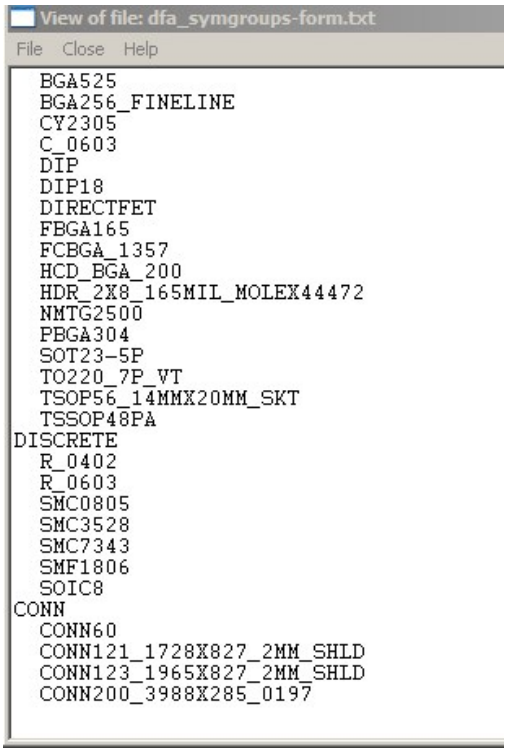


Figure 29 – Example of file

Adding Text (from a file)

To add text from a .txt file on disk:

1. Run the add text command: Add → Text.
2. Set the desired text options on the Control Panel "Options" tab.
3. Set the Active class to Board Geometry.
4. Set the Subclass to Dimension (or any other subclass).
5. Set Marker Size:
6. Set Rotation:
7. Set Text Block:
8. Set Text Justification:
9. Position the cursor/LMB at the location that you want the first line of text to start.
10. RMB and choose: Read from File.

Method to renumber all Ref Des from start

Use the fst_ref_des variable. You must ensure that the refdes digits in the Sequential Renaming section of the UI allow for the number of digits in the fst_ref_des variable. The variable is located in User Preference → Manufacturing → Silkscreen.

Why does the system react slowly when moving a comp?

Most likely because your large pin-count nets are scheduled Min Tree. Add a voltage property or RATSNEST_SCHEDULE = POWER_AND_GROUND to large pin-count nets like Vcc and GND.

Why does DRC update take hours to run?

Usually, the slower checking is a result of a large number of constraint areas and/or a large number of overlapping constraint areas. If that is not the case, then it might be the larger spacing values that you have to satisfy you high voltage requirements or simply fudge values entered as no-ops. Large-spacing values result in more neighboring items being examined for each item being checked.

Note: Review your extra-large spacing values. There could be a value of 999 entered. When reduced, DRC update can go from 1 hour to 3 minutes.

Is my database optimized for performance?

Performance Advisor provides suggestions to increase database performance. The command can be run from Tools → Database Check → Performance Advisor.

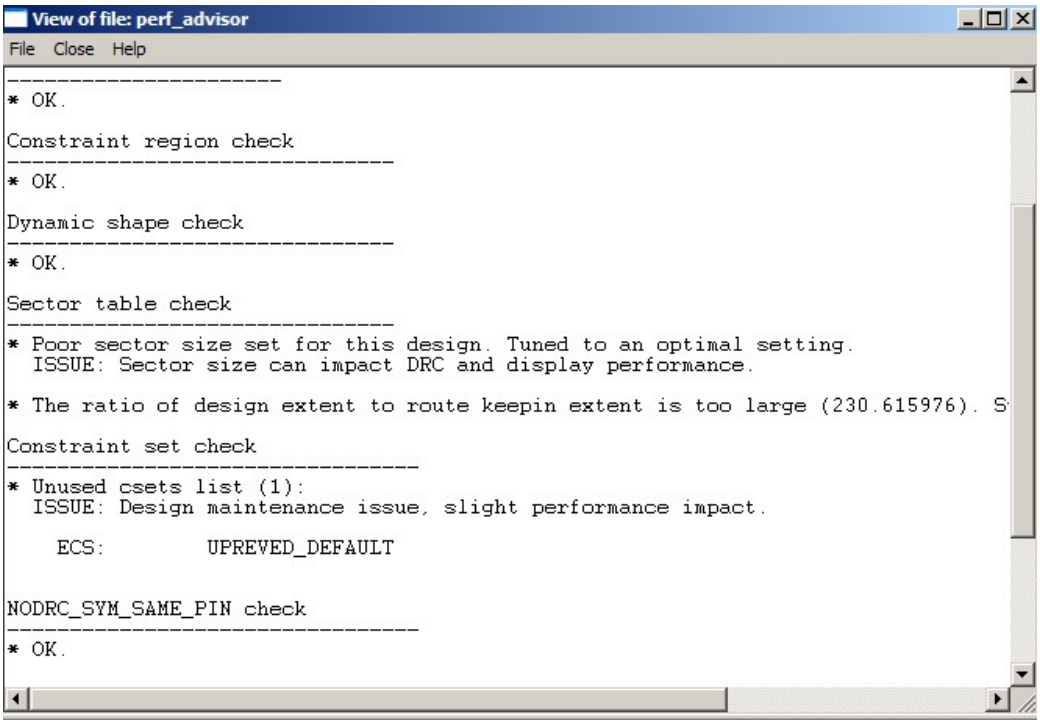


Figure 30

Read-only variables

A method for a site administrator to make variables read-only.

Add a “readonly” entry under the respective variable. In the following example, you wish to prevent the end user from modifying PSMPATH. The readonly psmpath command is added just below the "set psmpath" variable.

```
unset datatips_delay
set psmpath = D:\work\customers\IBM\ D:\work\marketing\Training\16.2\symbols\
readonly psmpath
set padpath = D:\work\marketing\Training\16.2\symbols\ D:\work\customers\IBM\
unset allegro_etch_length_on
set acon oldhlt = all
```

Figure 31

To verify in Allegro, type readonly at the command line prompt. The window lists variables that are read-only.

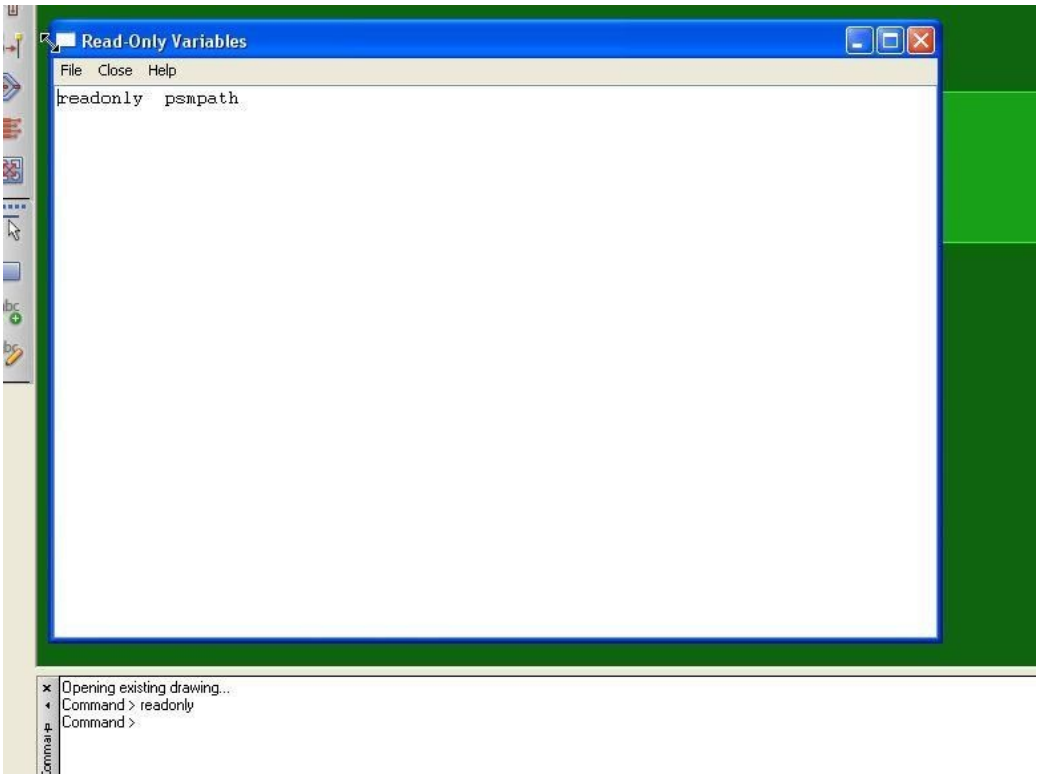


Figure 32

When an attempt is made to change psmpath in the User Preferences Editor, the following warning will appear:

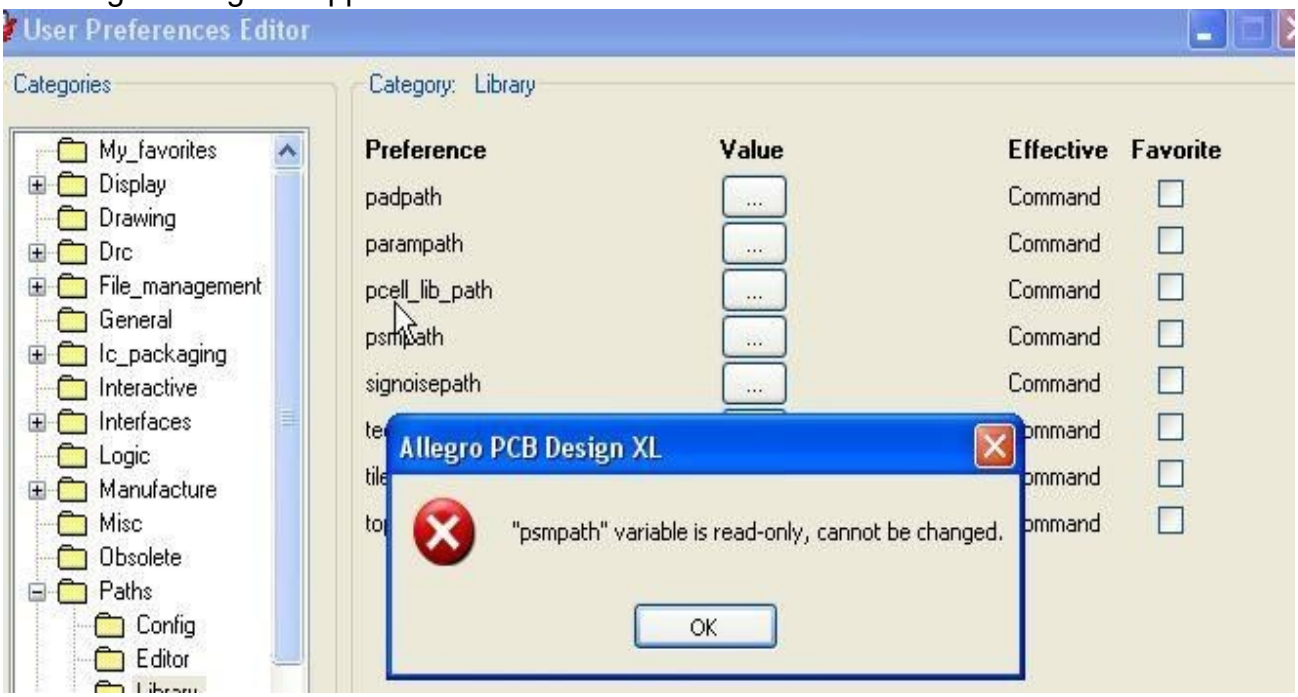


Figure 33

BBVIA GAP DRC reporting too many DRCs

Under normal circumstances, the 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 following example, the property value of "2" suppresses the DRC on the left side.

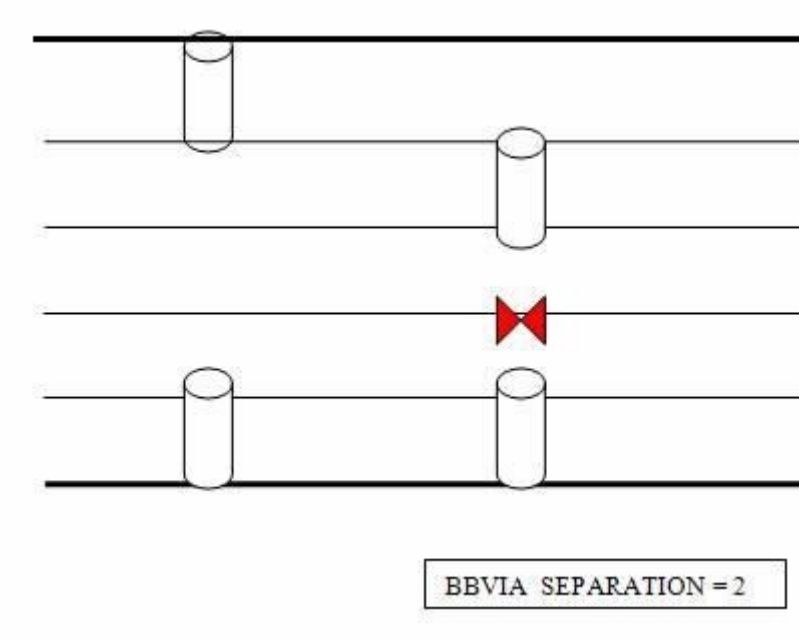


Figure 34

Same Net DRCs not being generated

It is most likely because your "DRC by-layer" setting is set to "False".

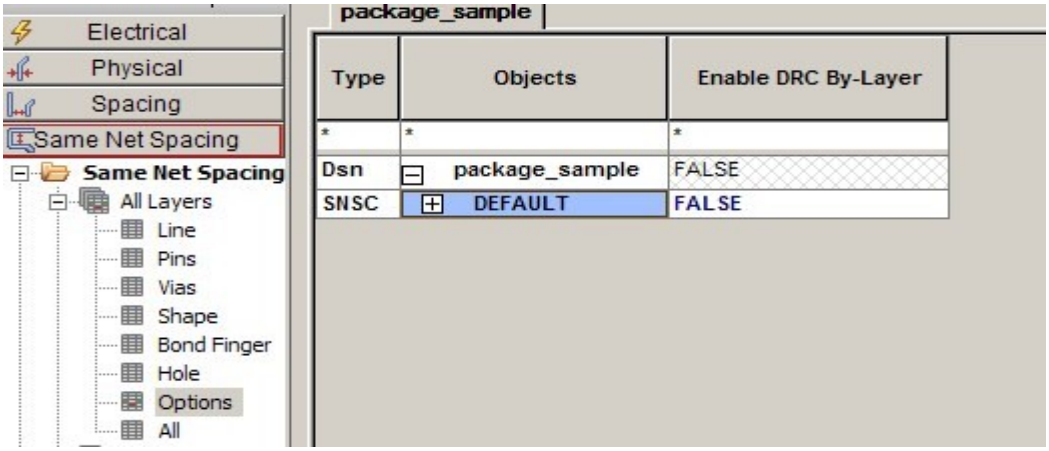


Figure 35

Working with Xnets

Create Xnets – Assign Espice models to the discrete devices from the Analyze → SI/EMI Sim → Model Assignment menu.

Prevent Xnets on an Instance basis - Assign the no_xnet_connection property at the component level.

Display Xnets – In CM, go to Object → Filter and enable "net" to see the nets of the Xnet.

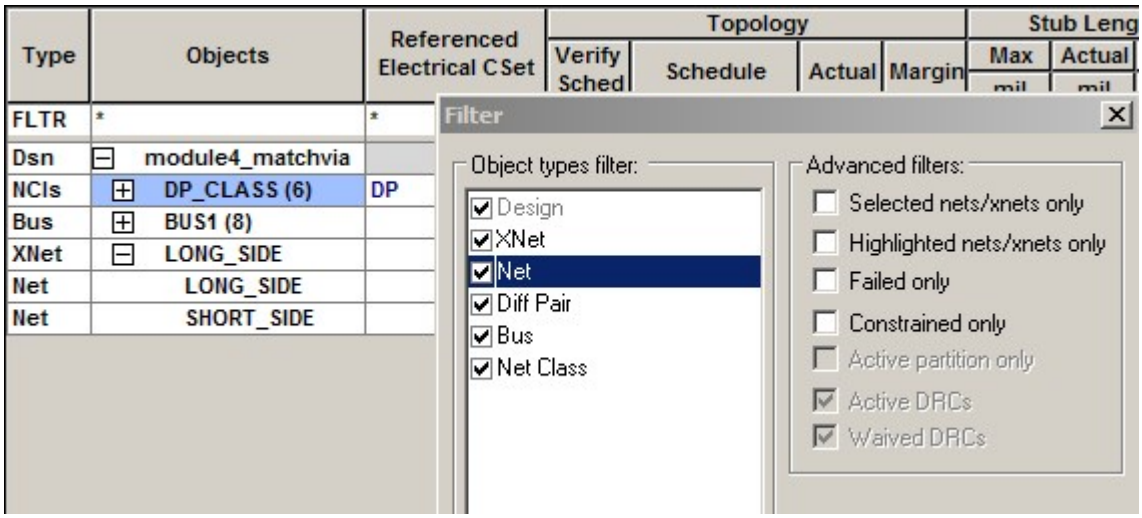


Figure 36

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

Change the Xnet Name

In CM, select the Xnet, and then use RMB → Rename

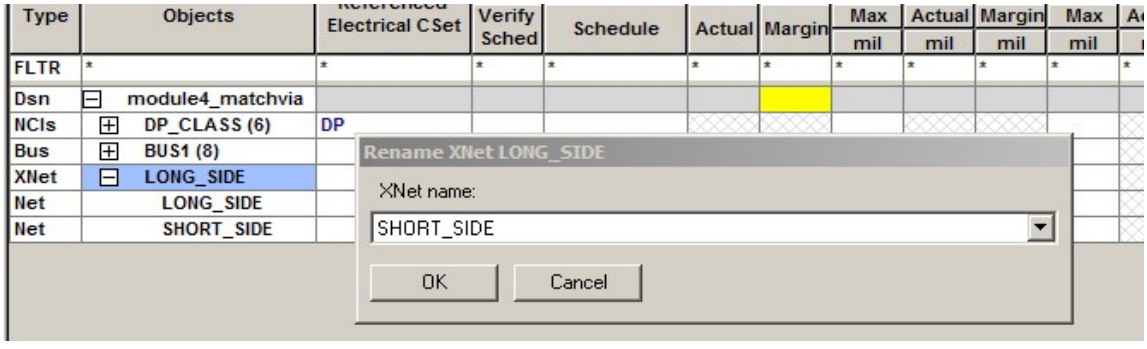


Figure 37

Gathering Diff Pair

Looking for tighter coupling at the gathering location?

Decrease the value of the “padentry_factor” variable located in ETCH category of the User Preferences Editor.

Figure 38

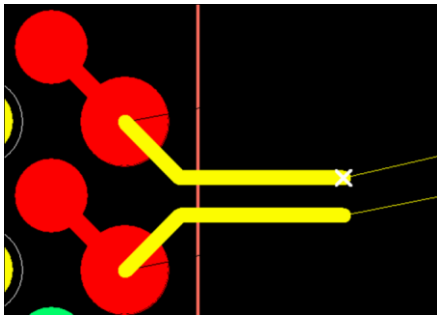


Figure 39 – Default

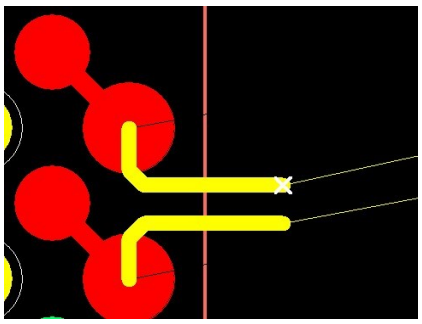


Figure 40 - Padentry set to 10

Routing a Tandem Diff pair

Pre-route each member as shown in the following figure, and then use Add Connect to route as a tandem pair. This assumes a Diff Pair constraint object has been created.

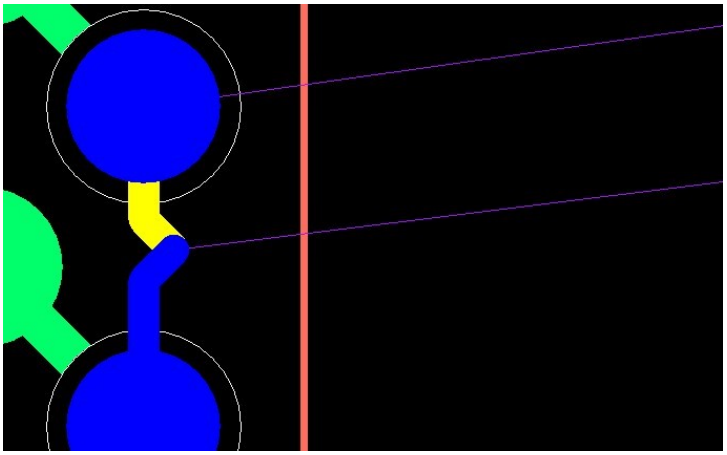


Figure 41

Displaying length meter for non-constrained nets

Set the “allegro_etch_length” variable located in User Preference → Route → Connect category.

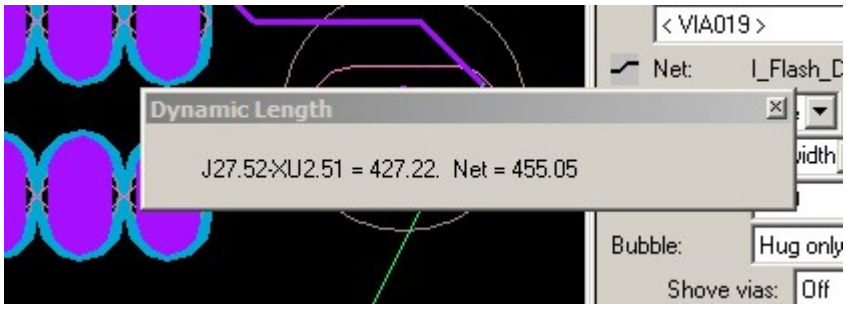


Figure 42

Constraint Manager

You wish to display propagation delay analysis for unrouted nets in CM. Enable the "unrouted" modes in Analyze → Analysis Modes → Options tab.

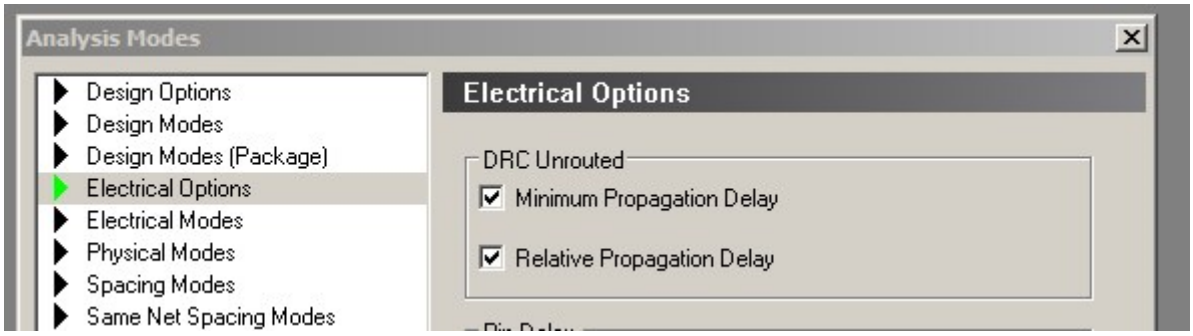


Figure 43

Sorting results from the worst case to best case margin

Double-click Margin Column Header, or RMB → Sort.

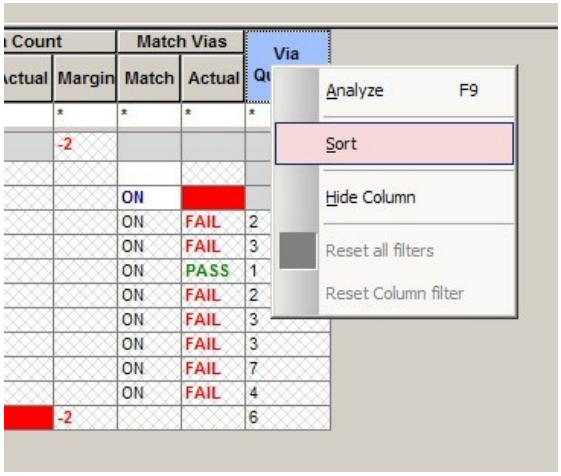


Figure 44

Displaying only the failed results

Click Object Header, RMB → Filter, and enable failed only, or simply click on the following icon:

Displaying only the nets that have constraints

Click Object Header, RMB → Filter, and enable constrained only, or simply click on the following icon:

Segregating sections of CM with divider lines

It, means identifying the transition between buses and matched groups.

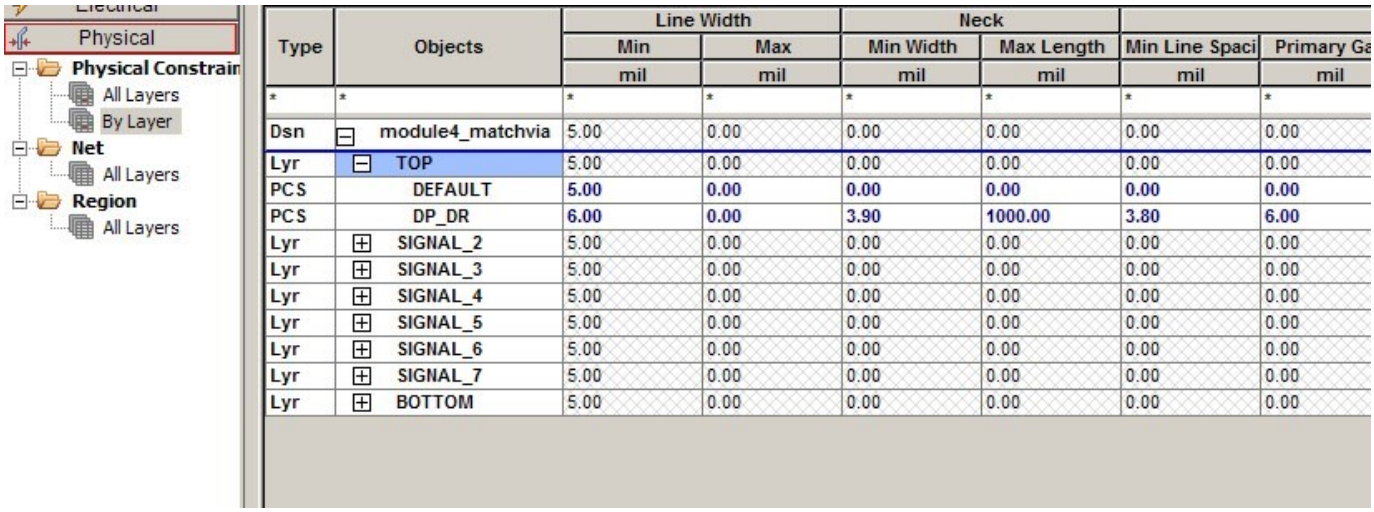
1. In CM, go to View → Options → Workbooks → Enable “Object Type Dividers”, or simply click on the following icon:

Type	Objects	Referenced Electrical CSet	Via Count			Match Vias		Via Quantity
			Max	Actual	Margin	Match	Actual	
*	*	*	*	*	*	*	*	*
Dsn	⊟ module4_matchvia				-2			
NCls	⊞ DP_CLASS (6)	DP						
Bus	⊟ BUS1 (8)					ON		
Net	AD0					ON	FAIL	2
Net	AD1					ON	FAIL	3
Net	AD2					ON	PASS	1
Net	AD3					ON	FAIL	2
Net	AD4					ON	FAIL	3
Net	AD5					ON	FAIL	3
Net	AD6					ON	FAIL	7
Net	AD7					ON	FAIL	4
XNet	⊞ LONG_SIDE				-2			6

Figure 45

You wish to change a constraint, for example, min line width across a certain layer of all the available PCSets.

Consider working in the “By Layer” worksheet to change the Top layer constraint across all the available Csets.



Type	Objects	Line Width		Neck			
		Min	Max	Min Width	Max Length	Min Line Spaci	Primary Ga
		mil	mil	mil	mil	mil	mil
*	*	*	*	*	*	*	*
Dsn	module4_matchvia	5.00	0.00	0.00	0.00	0.00	0.00
Lyr	TOP	5.00	0.00	0.00	0.00	0.00	0.00
PCS	DEFAULT	**5.00**	**0.00**	**0.00**	**0.00**	**0.00**	**0.00**
PCS	DP_DR	**6.00**	**0.00**	**3.90**	**1000.00**	**3.80**	**6.00**
Lyr	SIGNAL_2	5.00	0.00	0.00	0.00	0.00	0.00
Lyr	SIGNAL_3	5.00	0.00	0.00	0.00	0.00	0.00
Lyr	SIGNAL_4	5.00	0.00	0.00	0.00	0.00	0.00
Lyr	SIGNAL_5	5.00	0.00	0.00	0.00	0.00	0.00
Lyr	SIGNAL_6	5.00	0.00	0.00	0.00	0.00	0.00
Lyr	SIGNAL_7	5.00	0.00	0.00	0.00	0.00	0.00
Lyr	BOTTOM	5.00	0.00	0.00	0.00	0.00	0.00

Figure 46

Export to Excel

Constraint Manager can export to a native excel spreadsheet (Windows Only & you must have Excel installed).

In CM, go to Tools → Excel → Active Worksheet/Workbook.

Constraint resolution

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

Use Display → Constraint, and then window select the 2 elements to obtain spacing resolution. Next, select single element for physical resolution.

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

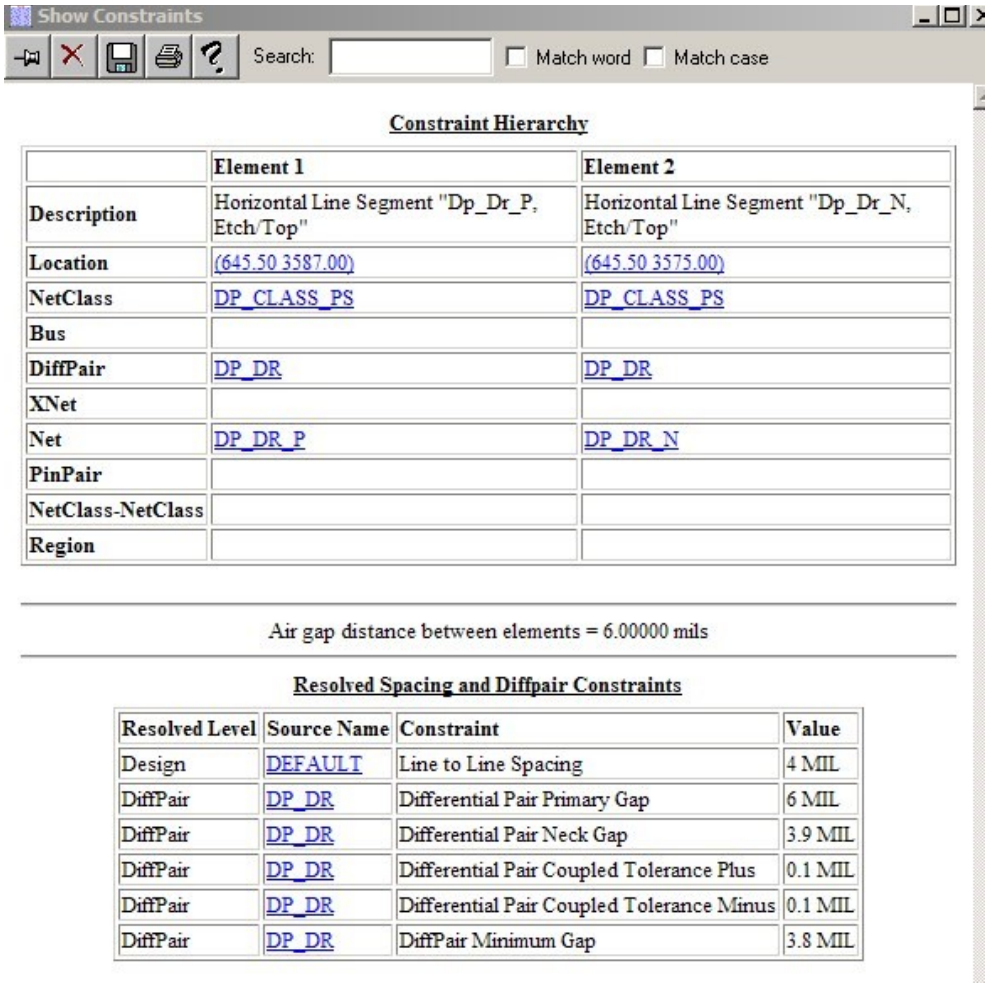


Figure 47

Assigning pins of dummy nets to net classes

In 16.3 (June 2010 ISR), enter the `cns_dummy_net` command or type `helpcmd` to get the complete Allegro command set.

Figure 48

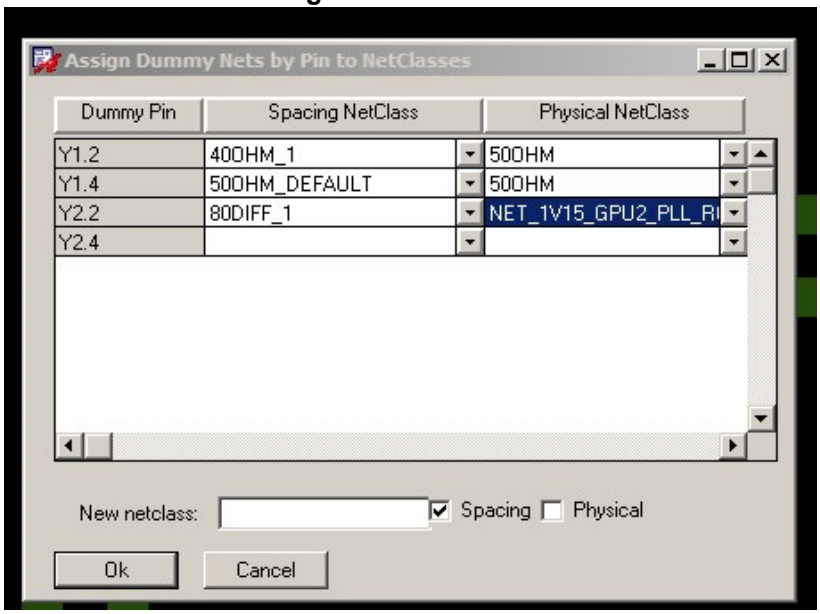


Figure 49

Preventing Shorts

In Allegro, It is possible to override a DRC with properties.

1. Make sure that the nets involved do not have the “NO_DRC” property on a pin(s).
2. Make sure there are no invalid “waived DRCs”.
3. Make sure your DRC modes are enabled. Be especially careful with newly-released DRCs (for example, Microvia-based) as those are disabled by default.
4. Run DBDoctor to keep the design in sync and up to date.
5. Make sure the plane layers in the cross section form agree with the artwork control form (negative or positive).
6. Make sure the padstacks are designed properly; both Antipads and thermals.
7. Include the IPC-D-356 netlist in your Fab Package. Ensure that the Fabricator does a netlist to Gerber compare.

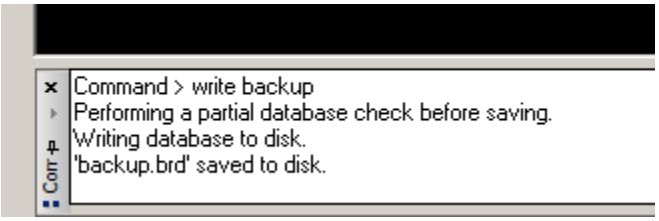
Suppressing SIGNOISE warnings and error messages

To suppress Signoise warning and error messages from appearing in the message window, set the ‘set sigsuppress ERROR WARNING’ variable.

Write vs. Save

You want to save the current board file you are working on as a new name. However, you 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.



Autosave the Database

Key variables to auto save the database.

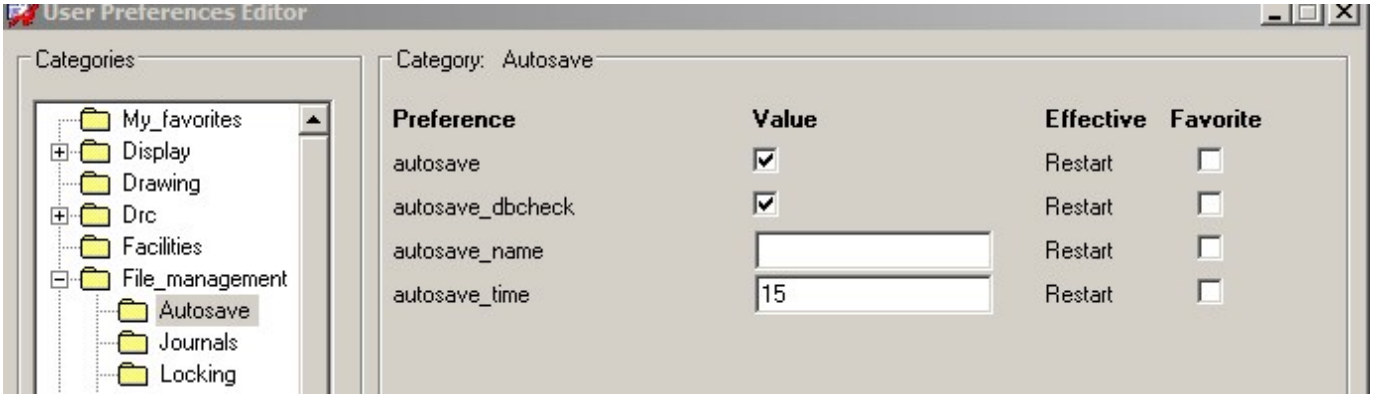


Figure 51

Reopen command

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

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

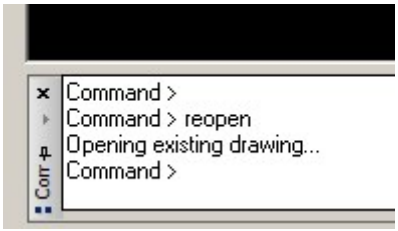


Figure 52

File Management (Artwork, Plots, Log Files, Reports)

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

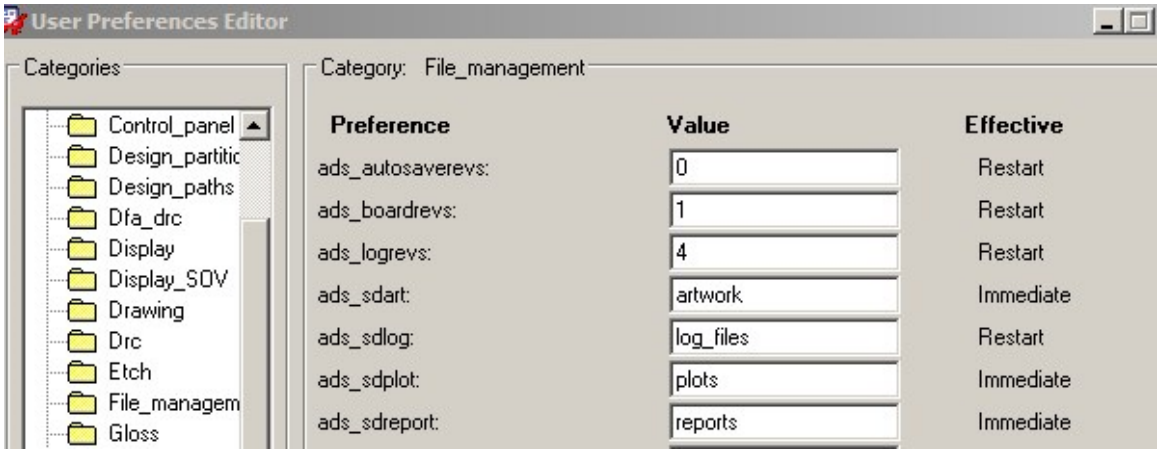


Figure 53

What is the earliest version of Allegro that can open my database?

- The command “Dbstat” will report the Allegro database revision and platform type.
- Dbstat supports the following file types (extensions):
- .brd .mdd .cio .sip .mdd .dpf .scf .pad .dra .mcm .psm .ssm .fsm, .bsm
- Type “Shell” in the Allegro command window to bring up a DOS window
- Type dbstate <file name>

Example:

```
tips_workshop.brd:   16.5  NT
```

Open File Manager from Allegro

Quick method to explore to your working directory:

Tools → Utilities → File Manager

Looking for a particular pad stack based on holesize?

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

1. Manufacturing → NC → Drill Customization → Library Drill Report
2. PadDesigner → Reports → Library Drill Report

Sort by column type by placing cursor in one of the cells, and then RMB → Sort by.

Figure 54

Testability → Multiple Probe-Type Support

The Testprep parameter form supports entries for multiple probe-types. Typically, the names are 100, 75 and 50 MIL. The names and center-to-center spacing is can be user-defined. To add a row in the form, select a Probe Type cell and RMB → Add.

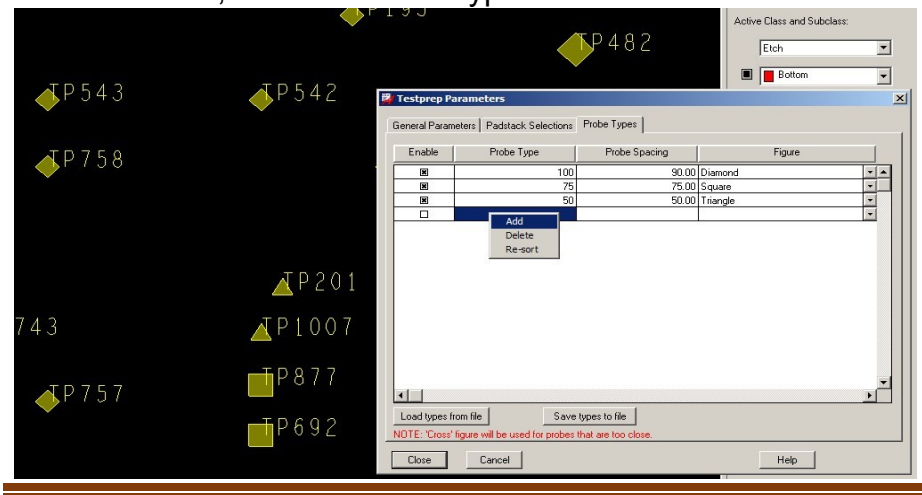


Figure 55

Drill Legend Support for INCHES and METRIC

Update your default .dlt file as shown here:

```
?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

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

Syntax: uprev_overwrite n where n = *.dra; *.psm; *.ssm

Example:
In a command window, enter `uprev_overwrite *.dra` to update all the .dra files to the current software level.

For more flexibility in upreving, in a DOS command window, type:

```
uprev -help
```

Note: Be sure to make a backup copy of your library before uprev.

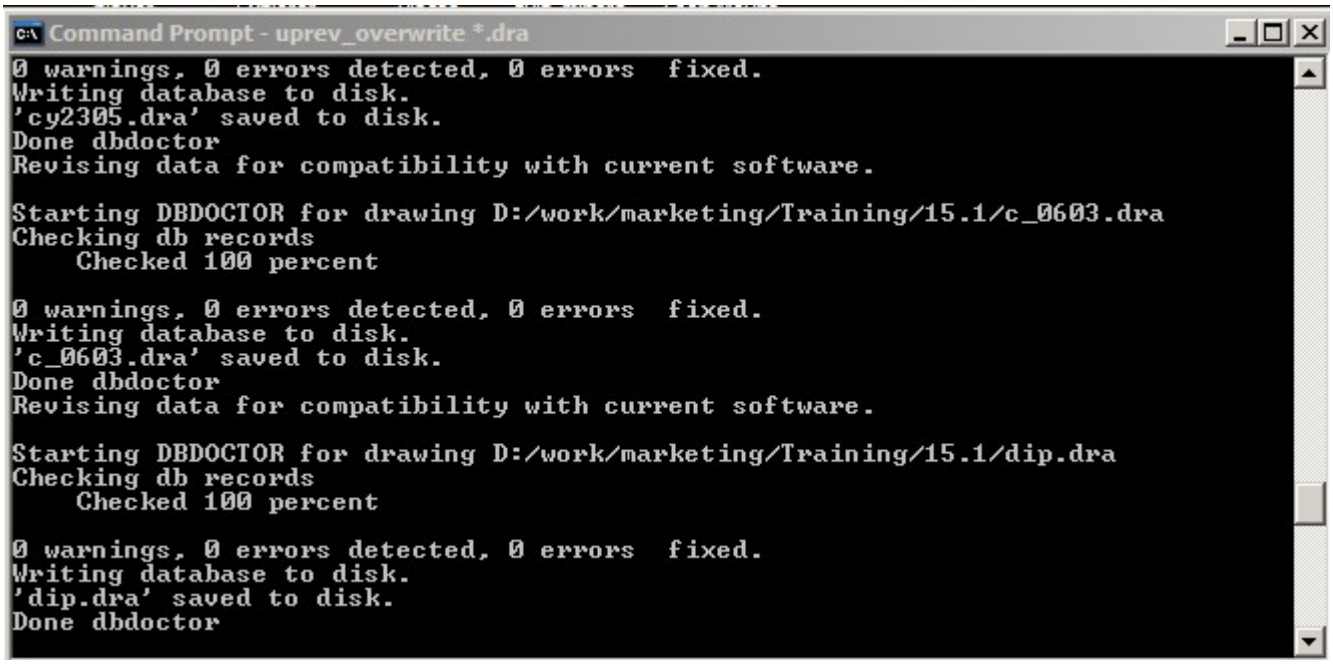


Figure 56

Troubleshooting: Running Allegro in Safe Mode

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

It disables loading at startup:

- 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)

UNIX: Currently it cannot disable X window resource settings.

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

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

http://www.cadence.com/community/forums/27.aspx?CMP=home

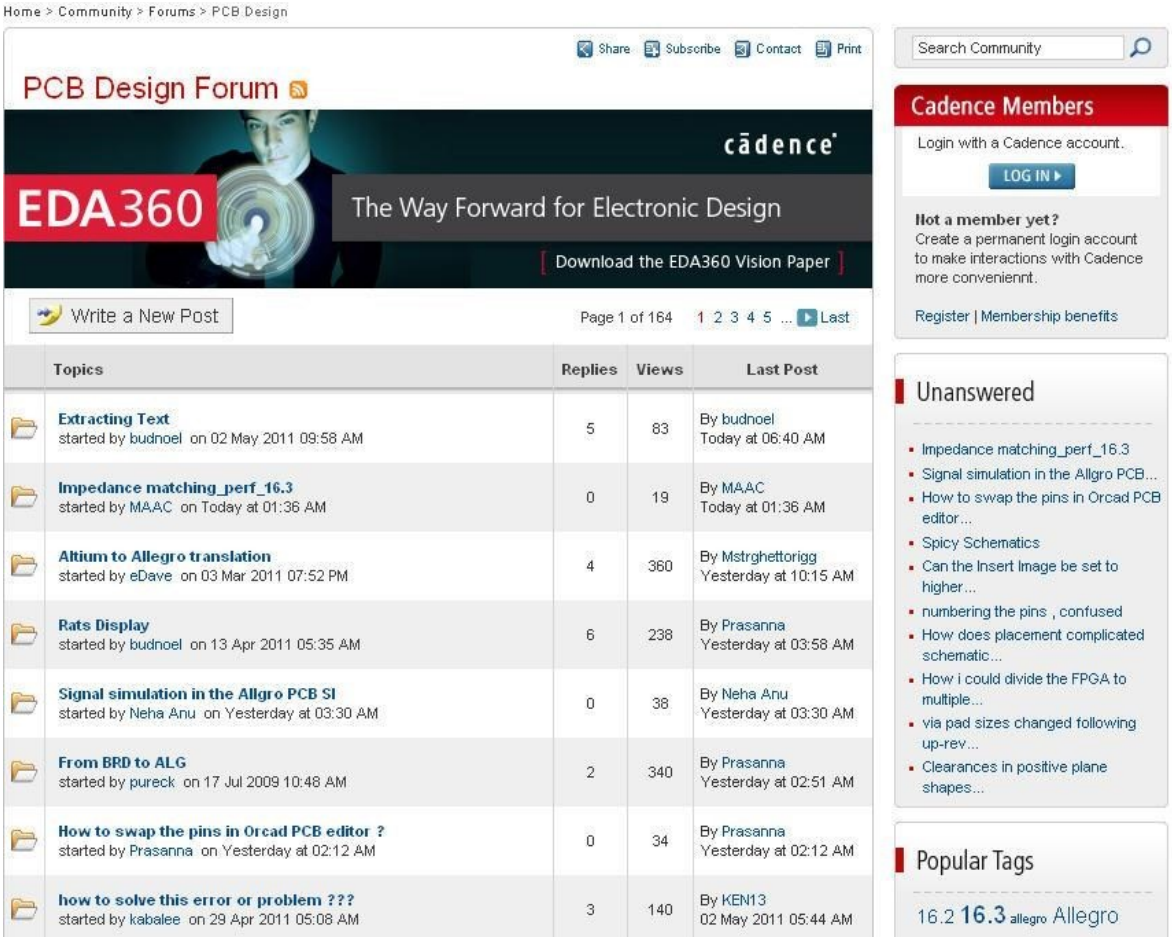
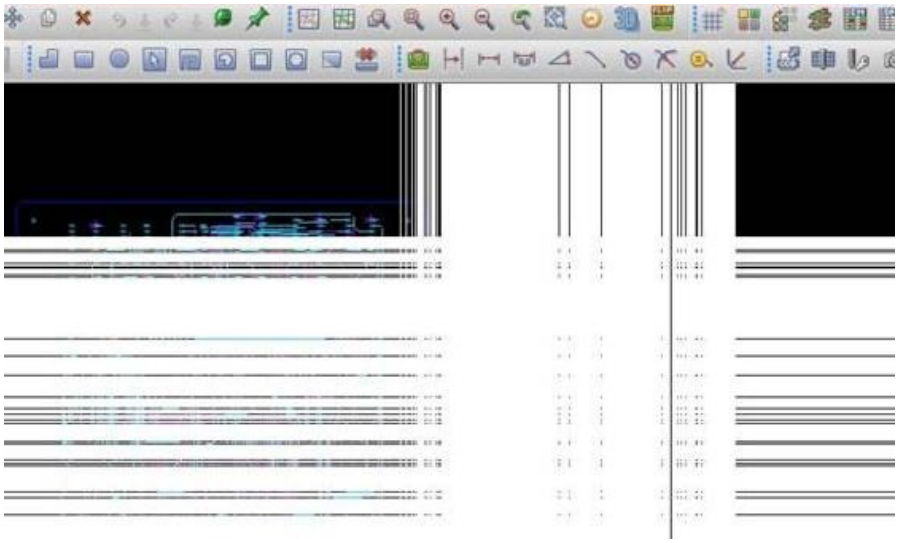


Figure 57

Windows 7: Graphic Issues



Try disabling Aero (desktop composition) for Allegro only:

1. Navigate to the executable or shortcut for the program you wish to disable desktop composition using Explorer.
2. Right-click on it, and select Properties.
3. Select the Compatibility tab and check Disable desktop composition, and then click on OK.
4. As always, ensure your graphic driver is up-to-date.

Support

Cadence Online Support provides access to support resources, including an extensive knowledge base, access to software updates for Cadence products, and the ability to interact with Cadence Customer Support. Visit *https://support.cadence.com*.

Feedback

Email comments, questions, and suggestions to content_feedback@cadence.com.