

# **PCB Editor Find Filter Object Definitions**

# **PCB Editor Find Filter Object Definitions**

This app note gives a brief explanation for the object types that are found within PCB Editor (OrCAD and Allegro). These items can be selected / not selected by using either the Find Filter (Docked / undocked menu on the right hand side of the screen by default) or via the Super Filter available from a right click - Super Filter – object type using the application modes in combination with most of the commands inside PCB Editor. The super filter precedes the Find Filter, for example if you select an object from the super filter the Find Filter will become "greyed out".



## **Object Types**

Groups - One or more objects linked together so that you can easily perform commands on them.

Comps - The combination of a symbol and logical description of a part.

Symbols - The physical description of a part, such as pins, part outline, and so on.

**Functions** – (Find Filter only). A logical unit of an electronic part such as an integrated circuit, also referred to as a gate.

Nets - The signal name associated with a component pin.

**Pins (Symbol Pin)** - Numbered electrical connection points (pads) on a symbol or component. Non-electrical pins on mechanical symbols or components do not have pin numbers.

Vias - The physical mechanism to traverse layers when connecting a net.

**Connect Lines, (Clines)** - A conductor trace associated with a net name. It begins and ends on a pin, via, or Tpoint.

Lines - A graphical line.

Bond wires – A Bond wire object used for Package on Board (Allegro) designs.

**Shapes** - A closed polygon. This shape may be used to represent internal power planes, keepout areas, keepin areas, and so on.

Voids /Cavities - Non-copper polygon or circle within an etch layer shape creating a cut-out.

**Cline Segs** - A portion of a cline. The segment is from one vertex (bend) point to the next vertex point (Route - Connect command).

Other Segs - Non-cline such as an arc, circle, and line (Add menu).

Figures - Pre-defined shapes that can be assigned to objects such as drill symbols. Found in padstack parameters.

DRC errors - Markers placed in the design to indicate errors after design rule checking (DRC) takes place.

Text - Annotation for design objects.

**Ratsnests** - Lines that show logical connections between two pins, clines, or vias in a design drawing. Elements connected by the same ratsnest line are part of the same net. The ratsnest shows the circuit logic and for Emitter Coupled Logic (ECL) circuits, the order in which pins are to be connected.

**Rat Ts** - Database objects used to insert a branch in nets' schedules at some point other than at a component pin. A rat T has a physical location that is often an approximate location for a 'T' or a via in the net's physical interconnect.

Off – (Super Filter only). In this mode the super filter is turned off and the Find Filter becomes available.

## **Find by Name**

This section allows users to locate items based on specific property types. Choose the required property or name type (like Net class for example) and then click on More which allows you to choose from the available Netclasses in the design. Once picked Apply or OK will select the relevant items in the main canvas window.

		🔡 Find by Name	e or Property			-		×
Find By Name  Property Net Symbol (or Pin) Device Type Symbol Type Property Bus Diff Pair Match Group Module Net Class Net Group Pin Pair BKO Group BKO Group	Name 🔻 More	Find by Name Object type: Available objects Name filter: Value filter: Value filter: Address Data Power_Group Rf Steve Steve1 Steve1 StevePh	e or Property Net Class	All-> <-All	Selected objects	_		×
RKO Group Region Xnet		Use 'selected ob	jects' for a deselection operati	on				
Generic Group		ОК	Cancel		Apply		He	ρ

## **Find by Query**

The Find by Query command, found on the Find pane, has been added to give the user total access to easily and quickly locate any and all design elements on the main canvas. Now, with a few simple clicks, and using the very

intuitive Find by Query dialog, users can create, save and recall time saving queries. The saved queries can also be re-used in new designs and can be used with all Edit commands.

🚼 Find by Query		_	
Objects Configure	Fields	Eilters	
Bond Fingers		Fillers	
Bond Wires			
Buses		->	
Cline Segments			
Clines			
Components			
Diffpairs			
DRC Errors			
Figures		AND	
Generic Groups		OR	
Line Segments			
Lines		<	>
Matched Groups	Matching Objects (0 of 0 Selected) Defer Selection	n	⊞ ©
Modules	······································		
Net Classes			
Net Groups			
Nets			
Pins			
Rat Bundles			
Ratsnests			
RatTs			
Regions			
RKO Groups			
Shapes			
Symbols			
Symbols RF			
Text			
Via Structures			
Vias			
Voids			
Xnets			
Load Query: Save Que	ry Clear Query Rerun Query	Export Result Close Cancel	Help

Queries are created by moving elements from the Objects tab to the Fields pane. As Objects are added to the Fields pane, they expand to reveal the attributes unique to them. Queries are then created and refined by moving attributes from the Fields pane to the Filters pane. As items are added to the Filters pane, items in the design that match the query are displayed in the Matching Objects pane. Elements can be moved from one pane to another in three possible ways:

- Double clicking on an element
- Drag and drop from one pane to another
- Using the left/right arrow buttons

Objects added to the Fields section may have multiple elements – all of which can be chosen to be part of the query. Available elements can be viewed or hidden simply by clicking on the small triangle to the left of the Object name.

✓ Sy	mbols	^
	Reference Designator	
	Component Class	
	Symbol Name	
	Symbol Type	
>	Location	
	Rotation Angle	
	Mirror Type	
~	Size	
	Length	
	Width	
	Thickness	
	Mounting Layer	
	Mounting Type	
	Pin Count	
	Group Owners	
>	Component Property	
~	Property	
	CLIP_DRAWING	
	DESCRIPTION	~



PCB Editor Find Filter Obj	ect Definitions		
💦 Filter	Setting		×
Operator:	equal to		-
Value:	*		
Selection	1		
Available	e Values: (filter available values)	Selected Values:	
C-136 CADE	7550-6 ^ NCE_LOGO		
CAPC	1005X56N 1608X86N	->	
CAPC	2012X70N	<-	
CAPC	2012X71N 2012X88N	>>	
CAPC	2012X100N		
CAPC	P3216X120N		
CAPM	P3216X180N V		
ОК	Cancel		

## Filters support wildcards (as shown below with "soic\*")

elds				Filt	Filters
<ul> <li>Symbo</li> <li>Ref</li> <li>Coi</li> <li>Syn</li> <li>Syn</li> <li>Loco</li> <li>Rot</li> <li>Mii</li> <li>Mii</li> <li>Sizz</li> <li>Thi</li> <li>Mo</li> <li>Mo</li> <li>Pin</li> <li>Gro</li> <li>Pro</li> </ul>	Is erence Designator mponent Class nbol Name nbol Type ation Angle ror Type ckness unting Layer unting Type Count Sup Owners mponent Property perty	1) 🗌 Defer Selection	-> <- AND OR	~	<ul> <li>Symbol</li> <li>AND</li> <li>Reference Designator equal to "*"</li> <li>Symbol Name equal to "SOIC*"</li> </ul>
Туре	Reference Designator				Symbol Name
Symbol	SW1	SOIC254P1077X800-8M			
Symbol	U3	SOIC127P600X175-8N			
Contrat	U7	SOIC127P1028X265-20N			
symbol					
Symbol	U22	SOIC127P1032X265-20AN			
Symbol Symbol Symbol	U22 U32	SOIC127P1032X265-20AN SOIC127P1028X265-20N			
Symbol Symbol Symbol Symbol	U22 U32 U35	SOIC127P1032X265-20AN SOIC127P1028X265-20N SOIC127P1028X265-20N			
Symbol Symbol Symbol Symbol Symbol	U22 U32 U35 U38	SOIC127P1032X265-20AN SOIC127P1028X265-20N SOIC127P1028X265-20N SOIC127P1032X265-20AN			
Symbol Symbol Symbol Symbol Symbol Symbol	U22 U32 U35 U38 U39	SOIC127P1032X265-20AN SOIC127P1028X265-20N SOIC127P1028X265-20N SOIC127P1032X265-20AN SOIC127P1032X265-20AN			

# Refined filters also support additional valid Operators – the example shown below is String Operators

🚼 Filter S	etting	×
Operator:	equal to	
Value:	equal to	1
Selection	contain not contain	

All Object Properties can be seen in the Fields pane

#### PCB Editor Find Filter Object Definitions

lds	Filters	
<ul> <li>Property</li> </ul>	∧ Symbol	
3D_MODEL	✓ AND	
ASSEMBLY_CONSTRAINT_SET	Reference Designator equal to "*"	
BACKDRILL_COMP_SIDE_ALLOWED	Symbol Name equal to "SOIC*"	
BACKDRILL_EXCLUDE		
BACKDRILL_MIN_PIN_PTH	<-	
BACKDRILL_OVERRIDE		
BACKDRILL_PRESSFIT_CONNECTOR		
CLIP_DRAW		
CLIP_DRAWING		
COMMENT		
DESCRIPTION		
DFA_DEV_CLASS		
DOGBONE_FANOUT		
DUAL_SIDED_COMPONENT	AND	
ECSET_MAPPING_TAG_UD		
EMB_VIA_CONNECT_PADSTACK	UK	
ETCH_TURN_UNDER_PAD		
FIXED		
IDF EXCLUDE		

### Matching Objects are viewed in a Table Format

Mato	hing Obj	ects (0 of 107 Selected	l) Defer Selection		1	ŝ
	Туре	Reference Designator	Symbol Name	Rotation Angle		^
1	Symbol	C1	CAPC1608X86N	90.000		
2	Symbol	C2	CAPC1608X86N	90.000		
3	Symbol	C15	CAPC1005X56N	90.000		
4	Symbol	C16	CAPC1005X56N	90.000		
5	Symbol	C20	CAPC1608X86N	90.000		
6	Symbol	C21	CAPC1608X86N	90.000		
7	Symbol	C31	CAPC1005X56N	90.000		
8	Symbol	C32	CAPC1005X56N	90.000		
9	Symbol	C33	CAPC1005X56N	90.000		
10	Symbol	C39	CAPC1608X86N	90.000		
11	Symbol	C42	CAPC1608X86N	90.000		
12	Symbol	C44	CAPC2012X100N	90.000		
				i		

Additional related Fields can be accessed and turned on by using the Table View Configuration icon (red square below). Returning to the Matching Objects view is done by clicking on the Table View Display COG in the top right hand corner.

Mato	hing Objects (0 of 10	7 Selected) Defer Selection			$\odot$
	Field	Sub-Field	Object Type	Show	^
1	Reference Designator		Symbols		
2	Component Class		Symbols		
3	Symbol Name		Symbols		
4	Symbol Type		Symbols		
5	Location	х	Symbols		
6	Location	Y	Symbols		
7	Rotation Angle		Symbols		
8	Mirror Type		Symbols		
9	Size	Length	Symbols		
10	Size	Width	Symbols		
11	Thickness		Symbols		

### Pin Count column added to Table View via the configuration settings

Matching Objects (0 of 107 Selected) Defer Selection Type Reference Designator Symbol Name Rotation Angle Pin Count Symbol C1 CAPC1608X86N 90.000 2 1 2 Symbol C2 CAPC1608X86N 90.000 2 3 Symbol C15 CAPC1005X56N 90.000 2 1.1.010 CARCINGEVECH 00.000 -

Matching Objects can be selected. Selected Objects count is located to the right of the Matching Objects pane title. Objects selected in the Matching Objects pane will also be selected (and zoomed in on) on the canvas.

Mato	hing Obj	ects (6 of 107 Selected	) Defer Selection		E E E E E E E E E E E E E E E E E E E	٥
	Туре	Reference Designator	Symbol Name	Rotation Angle	Pin Count	^
1	Symbol	C1	CAPC1608X86N	90.000	2	
2	Symbol	C2	CAPC1608X86N	90.000	2	
3	Symbol	C15	CAPC1005X56N	90.000	2	
4	Symbol	C16	CAPC1005X56N	90.000	2	
5	Symbol	C20	CAPC1608X86N	90.000	2	
6	Symbol	C21	CAPC1608X86N	90.000	2	
-	C 1 1	C21	CARCHOODVECH	00.000	2	

All objects can be selected in one of two ways – click on the upper left square in the Matching Objects spreadsheet or right click on any Matching Object and click on Select All.

Mate	hing Obj	ects (107 of 107 Select	ed) 🗌 Defer Select	tion						
	Туре	Reference Designator	Symbol Name	Rotation Angle	•	Mate	hing Ohi	acts (1 of 107 Salacted		
1	symbol	C1	CAPC1608X86N	90.000	2	Matt	aning Obj	ects (1 0) 107 Selected		
2	Symbol	C2	CAPC1608X86N	90.000	2		Tune	Reference Designator	Symbol Name	Rot
3	Symbol	C15	CAPC1005X56N	90.000	2		type	Reference Designator	Symbol Mame	NUG
4	Symbol	C16	CAPC1005X56N	90.000	2	1	Symbol	C1	CAPC1608X86N	90.00
5	Symbol	C20	CAPC1608X86N	90.000	2	2	Symbol	C2	CAPC	
6	Symbol	C21	CAPC1608X86N	90.000	2	3	Symbol	C15	CAPC Select A	AII .
7	Symbol	C31	CAPC1005X56N	90.000	2	4	Symbol	C16	CARC1005X56N	00.00
8	Symbol	C32	CAPC1005X56N	90.000	2	-	Symbol	C10	CAP C1609X96N	00.0

If the PCB design changes while the query dialog is active, the present query will display a notice indicating that changes have occurred and that the present query is no longer valid. Rerunning the query will bring the Matching Objects results back in sync.

jects Configure	Field	ls					Filters	
Bond Fingers Bond Wires Buses Cline Segments Cimes Components Diffpairs DRC Errors Figures Generic Groups Line Segments Lines		Symb Symb > Locat Rotati Mirro > Size Thick Hor Etcr Group > Comp > Comp	ol Name ol Type ion <b>on Angle</b> r Type izontal Line Segment Wi John John Vet name: N8851 John O Owners onent Property rtv	idth: 0.200 88		-> <- AND OR	<ul> <li>Symbol</li> <li>AND</li> <li>Reference Designator equal to """</li> <li>Symbol Name equal to ""</li> <li>Rotation Angle = "90.000"</li> </ul>	
Matched Groups Modules	The	board ha	as changed since the qu	iery was last run. P	Please Rerun	Query.		
Net Classes		Туре	Reference Designator	Symbol Name	Rotation A	ngle	Pin Count	
Net Groups	1	Symbol	C1	CAPC1608X86N	90.000	2		
Nets	2	Symbol	C2	CAPC1608X86N	90.000			
rins Pat Rundles	3	Symbol	C15	CAPC1005X56N	90.000	2		
Rat Bundles	4	Symbol	C16	CAPC1005X56N	90.000	2		
Addites to	5	Symbol	C20	CAPC1608X86N	90.000	2		
actions	6	Symbol	C21	CAPC1608X86N	90.000	2		
2KO Groups	7	Symbol	C31	CAPC1005X56N	90.000	2		
hanes	8	Symbol	C32	CAPC1005X56N	90.000	2		
wmbols	9	Symbol	C33	CAPC1005X56N	90.000	2		
Symbols RE	10	Symbol	C39	CAPC1608X86N	90.000	2		
ievt	11	Symbol	C42	CAPC1608X86N	90.000	2		
/ia Structures	12	Symbol	C44	CAPC2012X100N	90.000	2		
	13	Symbol	C45	CAPC2012X100N	90.000	2		
/ias		Symbol	C47	CAPC2012X100N	90.000	2		
/ias /oids	14					-		
vias Voids Knets	14	Symbol	C48	CAPC2012X100N	90.000	2		

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

Other Trademarks

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

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