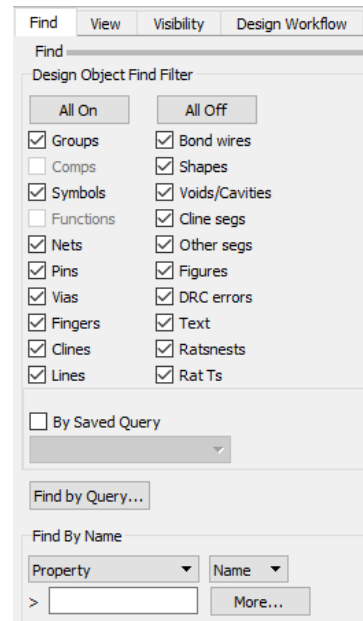
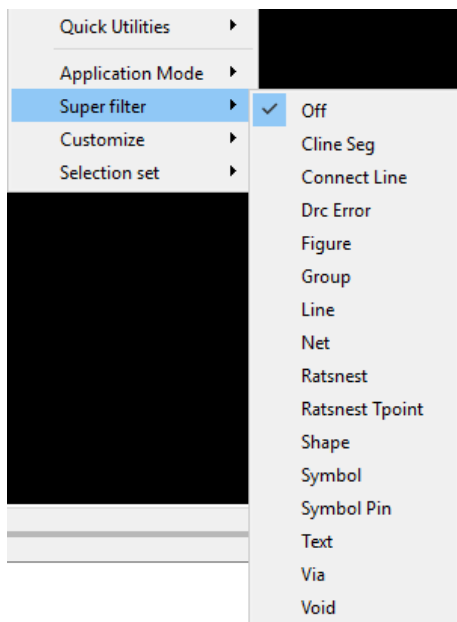




PCB Editor Find Filter Object Definitions

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.

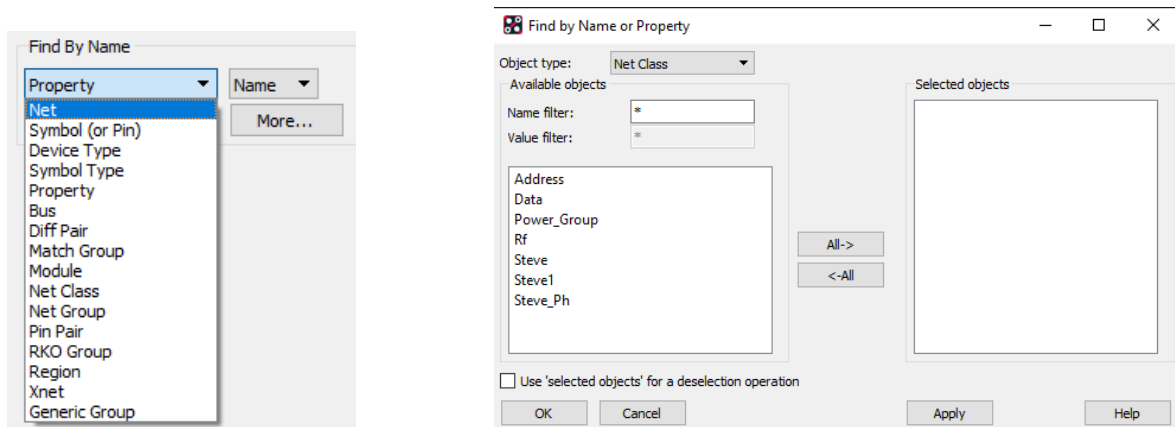
Ratsnests - Lines that show logical connections between two pins, clines, or vias in a design drawing. Elements connected by the same ratsnest line are part of the same net. The ratsnest shows the circuit logic and for Emitter Coupled Logic (ECL) circuits, the order in which pins are to be connected.

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

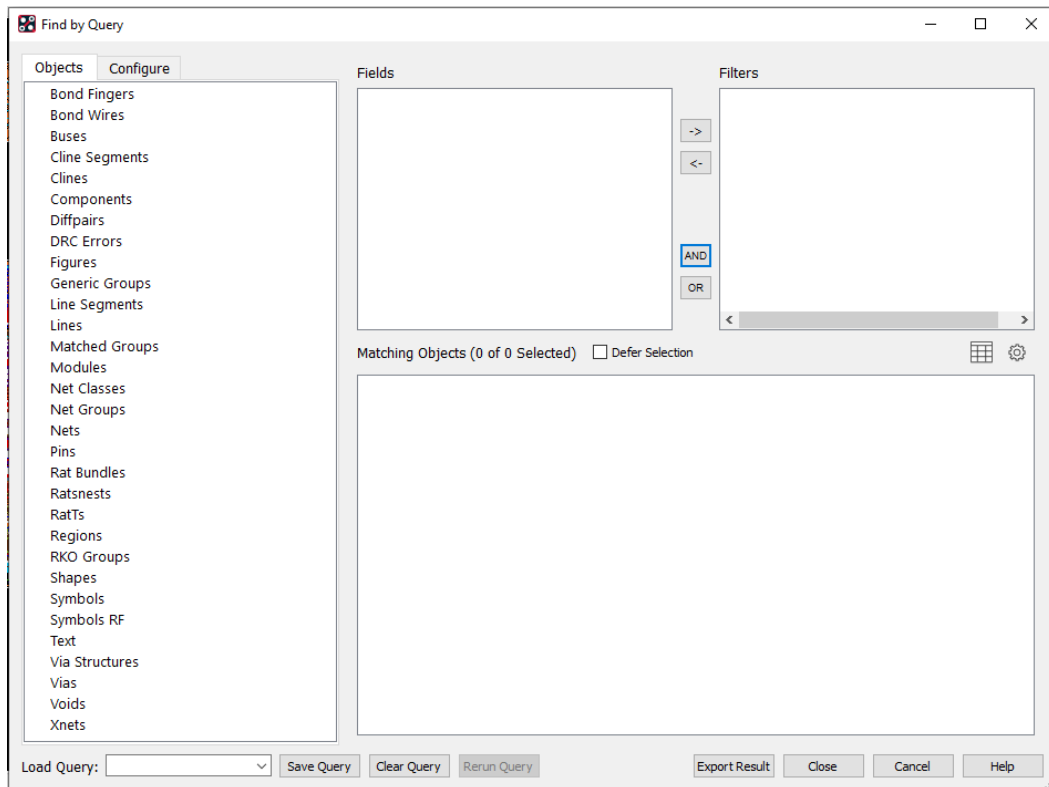
This section allows users to locate items based on specific property types. Choose the required property or name type (like Net class for example) and then click on More which allows you to choose from the available Netclasses in the design. Once picked Apply or OK will select the relevant items in the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



Find by 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

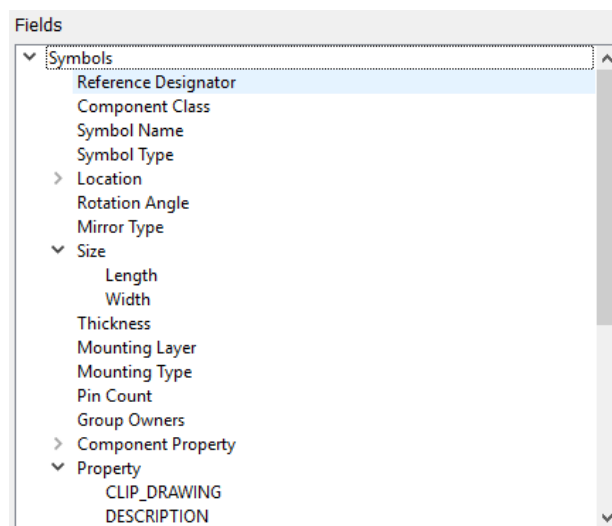
intuitive Find by Query dialog, users can create, save and recall time saving queries. The saved queries can also be re-used in new designs and can be used with all Edit commands.



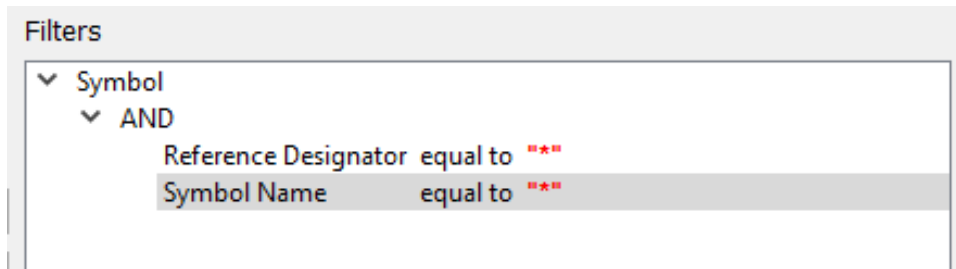
Queries are created by moving elements from the Objects tab to the Fields pane. As Objects are added to the Fields pane, they expand to reveal the attributes unique to them. Queries are then created and refined by moving attributes from the Fields pane to the Filters pane. As items are added to the Filters 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**

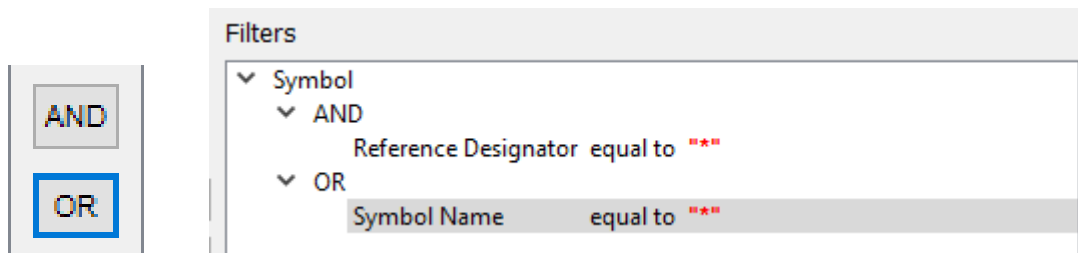
Objects added to the Fields section may have multiple elements – all of which can be chosen to be part of the query. Available elements can be viewed or hidden simply by clicking on the small triangle to the left of the Object name.



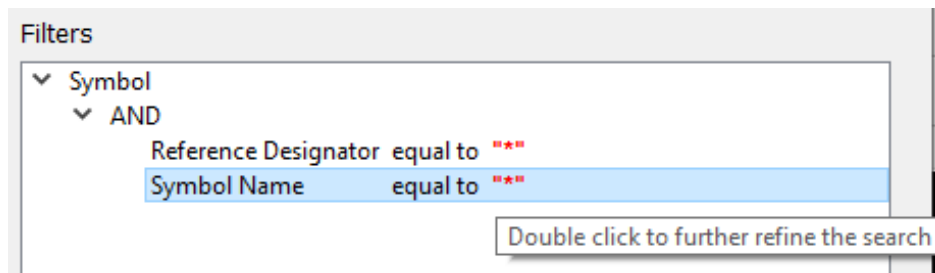
Queries are created once elements are added to the Filters pane. The example below equates to Symbols that have a Ref Des equal to "*" and a Package Symbol Name equal to "*"



Selecting the Or button will create a different expression as shown below



Added Filter default to "*" - which equates to all possibilities. The bold "*" and red text is a visual indicator that additional refinements can be made. A hover over pop up has also been added as a reminder.

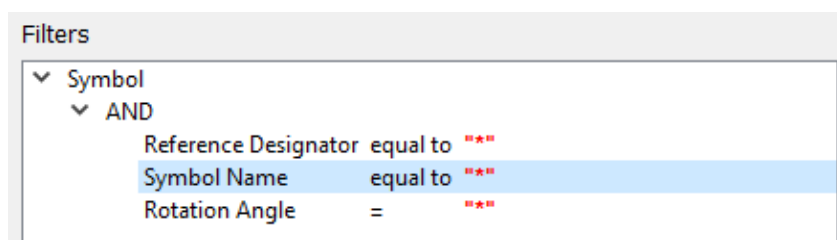


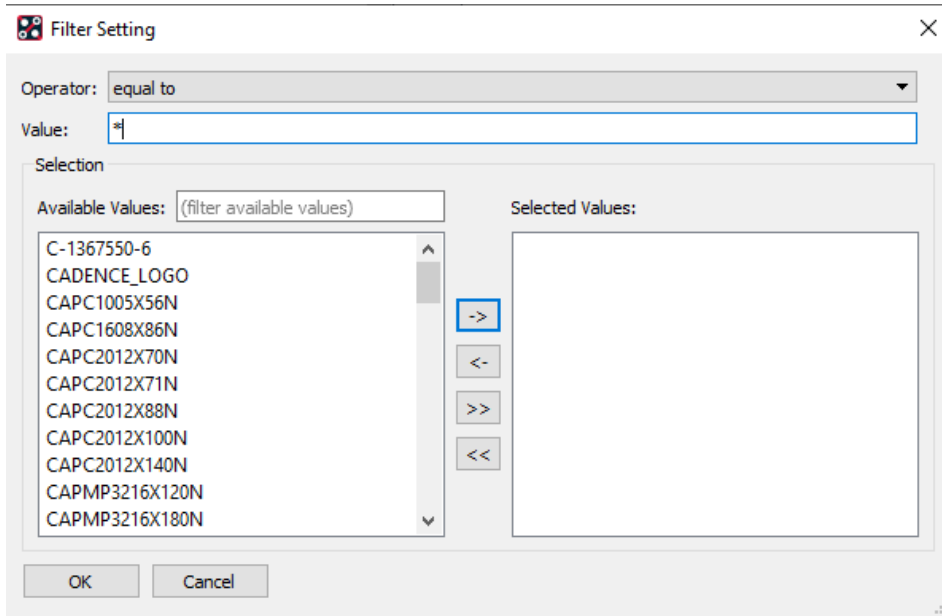
Current results that match the query can be found in the Matching Objects pane

Matching Objects (0 of 355 Selected) Defer Selection

	Type	Reference Designator	Symbol Name
1	Symbol	C1	CAPC1608X86N
2	Symbol	C2	CAPC1608X86N
3	Symbol	C3	CAPC2012X71N
4	Symbol	C4	CAPC2012X71N

- Query Refinements can be done by double clicking on any red bolded text

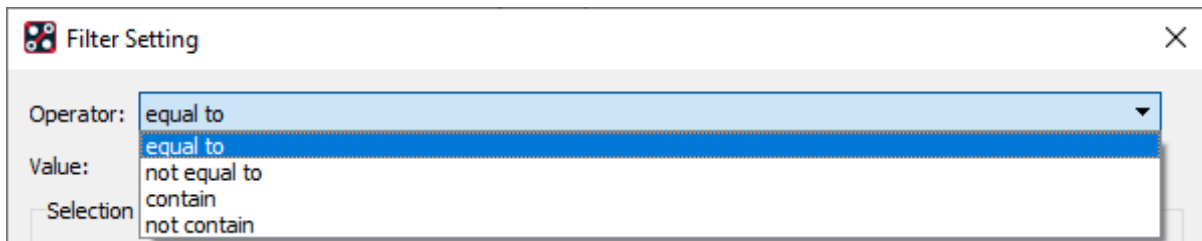




Filters support wildcards (as shown below with "soic*")

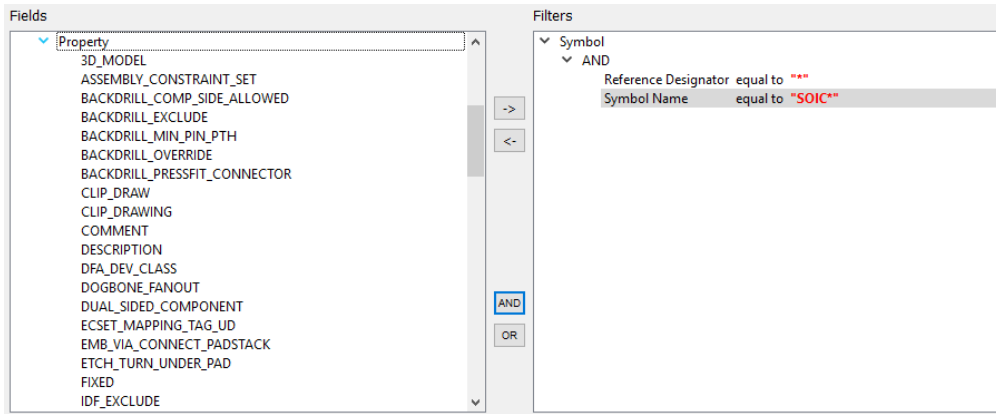
Type	Reference Designator	Symbol Name
1	Symbol SW1	SOIC254P1077X800-8M
2	Symbol U3	SOIC127P600X175-8N
3	Symbol U7	SOIC127P1028X265-20N
4	Symbol U22	SOIC127P1032X265-20AN
5	Symbol U32	SOIC127P1028X265-20N
6	Symbol U35	SOIC127P1028X265-20N
7	Symbol U38	SOIC127P1032X265-20AN
8	Symbol U39	SOIC127P1032X265-20AN
9	Symbol U48	SOIC127P1028X265-20N

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



All Object Properties can be seen in the Fields pane

PCB Editor Find Filter Object Definitions



Matching Objects are viewed in a Table Format

Matching Objects (0 of 107 Selected) Defer Selection

	Type	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

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.

Matching Objects (0 of 107 Selected) Defer Selection

	Field	Sub-Field	Object Type	Show
1	Reference Designator		Symbols	<input checked="" type="checkbox"/>
2	Component Class		Symbols	<input type="checkbox"/>
3	Symbol Name		Symbols	<input checked="" type="checkbox"/>
4	Symbol Type		Symbols	<input type="checkbox"/>
5	Location	X	Symbols	<input type="checkbox"/>
6	Location	Y	Symbols	<input type="checkbox"/>
7	Rotation Angle		Symbols	<input checked="" type="checkbox"/>
8	Mirror Type		Symbols	<input type="checkbox"/>
9	Size	Length	Symbols	<input type="checkbox"/>
10	Size	Width	Symbols	<input type="checkbox"/>
11	Thickness		Symbols	<input type="checkbox"/>

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
1	Symbol	C1	CAPC1608X86N	90.000	2
2	Symbol	C2	CAPC1608X86N	90.000	2
3	Symbol	C15	CAPC1005X56N	90.000	2

PCB Editor Find Filter Object Definitions

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.

Matching Objects (6 of 107 Selected) Defer Selection

	Type	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

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.

Matching Objects (107 of 107 Selected) Defer Selection

	Type	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
7	Symbol	C31	CAPC1005X56N	90.000	2
8	Symbol	C32	CAPC1005X56N	90.000	2

Matching Objects (1 of 107 Selected) Defer Selection

	Type	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

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.

The board has changed since the query was last run. Please Rerun Query.

	Type	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
7	Symbol	C31	CAPC1005X56N	90.000	2
8	Symbol	C32	CAPC1005X56N	90.000	2
9	Symbol	C33	CAPC1005X56N	90.000	2
10	Symbol	C39	CAPC1608X86N	90.000	2
11	Symbol	C42	CAPC1608X86N	90.000	2
12	Symbol	C44	CAPC2012X100N	90.000	2
13	Symbol	C45	CAPC2012X100N	90.000	2
14	Symbol	C47	CAPC2012X100N	90.000	2
15	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.