



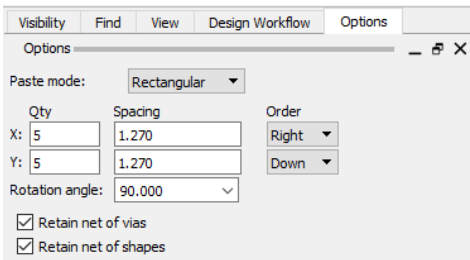
Introduction

Many designs today require signals to be shielded. This can be on a net by net basis where the net is surrounded by an array of vias connected to a ground plane below, or the board profile may well have an array of grounding vias surrounding the complete profile to improve EMC performance. On a more general example, if the design is using multiple power and ground planes, designers require the ability to “stitch” the planes together using via arrays. Although some of these basic functions are available using an OrCAD PCB Designer Standard License (stitching vias) the other more advanced operations require the OrCAD PCB Designer Professional license.

“Stitching” Vias.

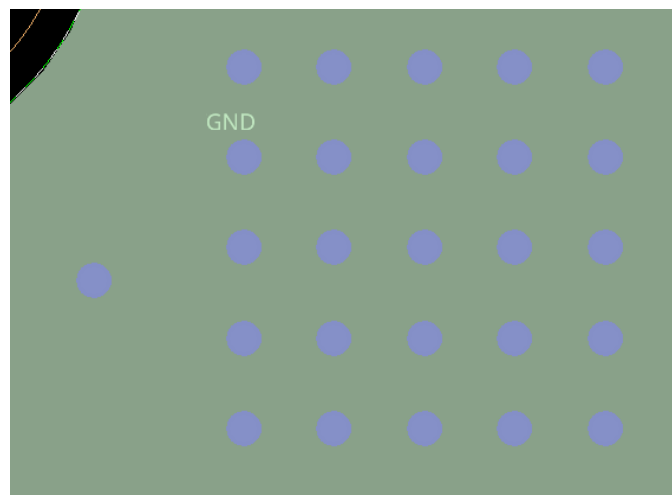
To add stitching vias using an OrCAD PCB Designer Standard license use Route > Add Connect, go to the location for the first via and double click the left mouse button (or right click > Add Via), This action adds a single via. It is important to note using this process to ensure that the actual via required is defined in Constraint Manager and that the active plane for connection (in this example GND) is the first layer visible. The net the via is connected to is the first plane found in the stackup so if you have a VCC plane on layer 2 the via added is connected to VCC. To alter this mode just display the layers you wish to connect from and that will be the default net used. There are many other ways to control this function and also please remember that this example is for a through via across all layers. Should your vias be defined as blind / buried you may need to define which via and layer you wish to connect from / to.

To create an array of stitching vias use the Edit > Copy command, once invoked (and the object you wish to copy is selected) look at the Options fold out menu for further settings:-



You have options for a Rectangular or Polar Array, then Qty X and Y, spacing between vias and array direction and angle. For this example the copy options have been defined as shown. Please ensure that the Retain net of vias checkbox is checked, this ensures that any vias copied use the same net as the source via.

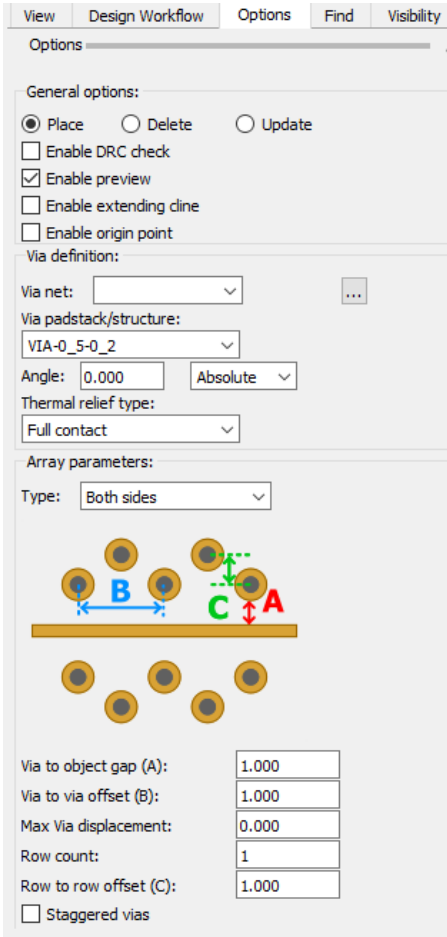
Once the settings are defined pick the source via and a new via is attached to the mouse, pick the location and the array of vias (in this case 25 vias) are added.



For users using an OrCAD PCB Designer Professional or an Allegro PCB Designer license and above there is a menu command Place > Via Arrays. The Options pane changes and allows you to define the type of via array you require.

Via Array

To create a Matrix array use Place > Via Array, then look at the options pane.



Once via array parameters are defined, you can place, update, and delete different array types by selecting relevant dynamic controls, available in the Options pane. The command also provides options to enable the preview mode while adding or updating a via array. Modifying any array parameter or changing the cursor location, changes the preview dynamically. Hence providing more control to adjust the settings before actually placing a via array.

You can add both vias or structures using the same command and in various patterns to the designs. A graphical view of each via array pattern is displayed in the Options pane. Swapping between the array types toggles all appropriate settings and graphics.

You can place arrays around the following objects. Depending on the array type, the Find filter enables the valid objects:- Pins, Vias, Clines, Cline Segments, Symbols and Shapes.

Placing a group of vias on a dynamic shape have the following limitations:

- You can only use an etch/conductor shape.
- The tool ignores void boundaries.
- Vias do not generate if the etch/conductor pad is partially outside the shape.
- DRC rules control all types of via arrays, and any vias in the array do not generate if they cause DRC errors.

You can insert a group of vias in an empty area of a specified region in a matrix pattern or around a shape as a boundary. The command provides following array types to add:

Boundary Arrays

Single Sided – Adds an array along one side of one or more selected object

Both Sides – Adds an array on both sides of one or more selected objects

Centered – Adds an array centered on one or more selected objects

Between – Adds an array between all selected objects that are parallel to each other

Surrounding – Adds an array surrounding the selected objects

Radial – Adds a circular pattern of vias or structures around one or more selected objects

Matrix Arrays

Across Board – A matrix of vias or structures is added filling the board outline

Across Shape – Matrices are added filling one or more selected shapes

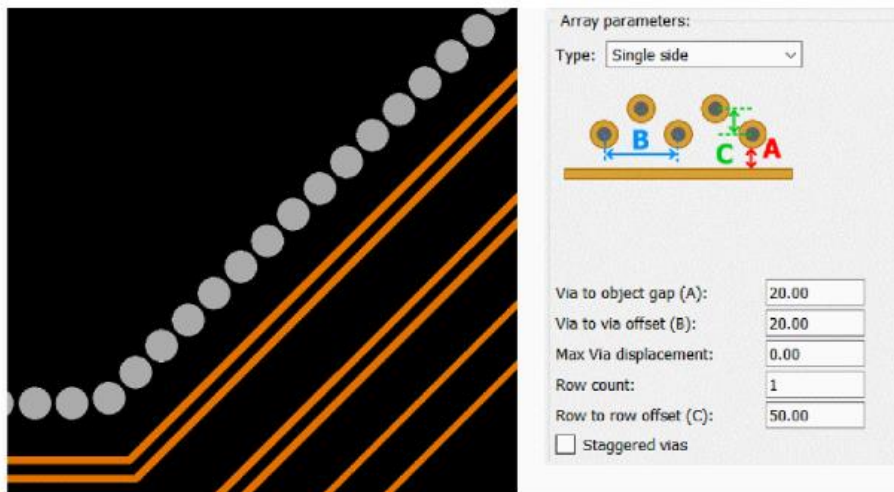
Across Windowed Area – A matrix of vias or structures is added to a windowed area

Via Array Examples

Adding a via or structure array in the design can be done by selecting the Place > Via Array command and then set the array type in the Options pane. You can add, modify and delete different types of arrays without exiting the command. When you place a via array within a given area, auto voids are produced. If you do not want to place vias at the same position or in an overlapped position, turn the Same net DRC on.

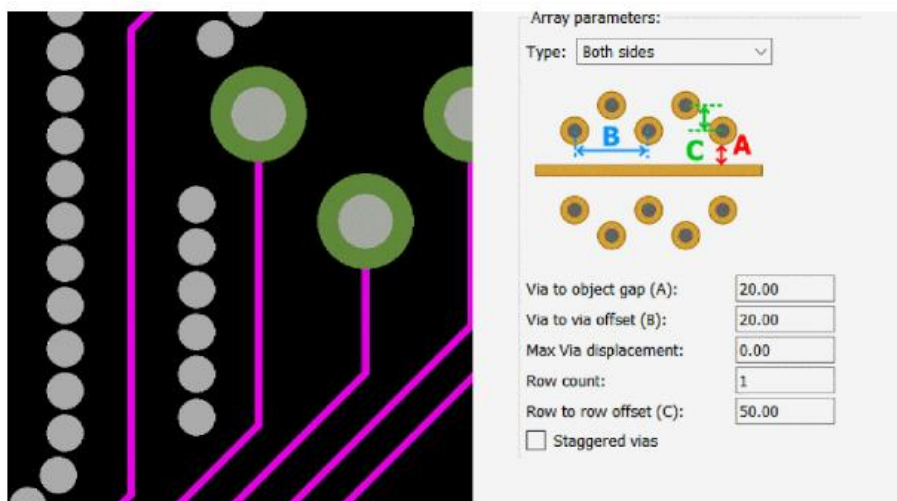
Single Side

Select a cline or shape to place via array on one side.



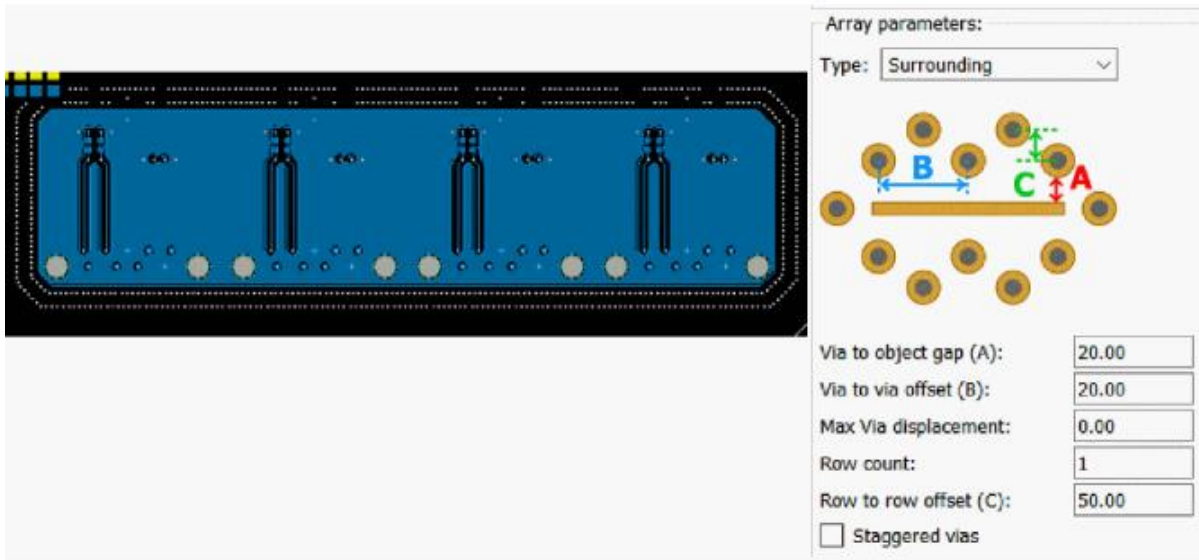
Both Sides

Via arrays can be added to both sides of a selected cline or shape objects. An array is not added if enough space is not available on any of the sides of the object.



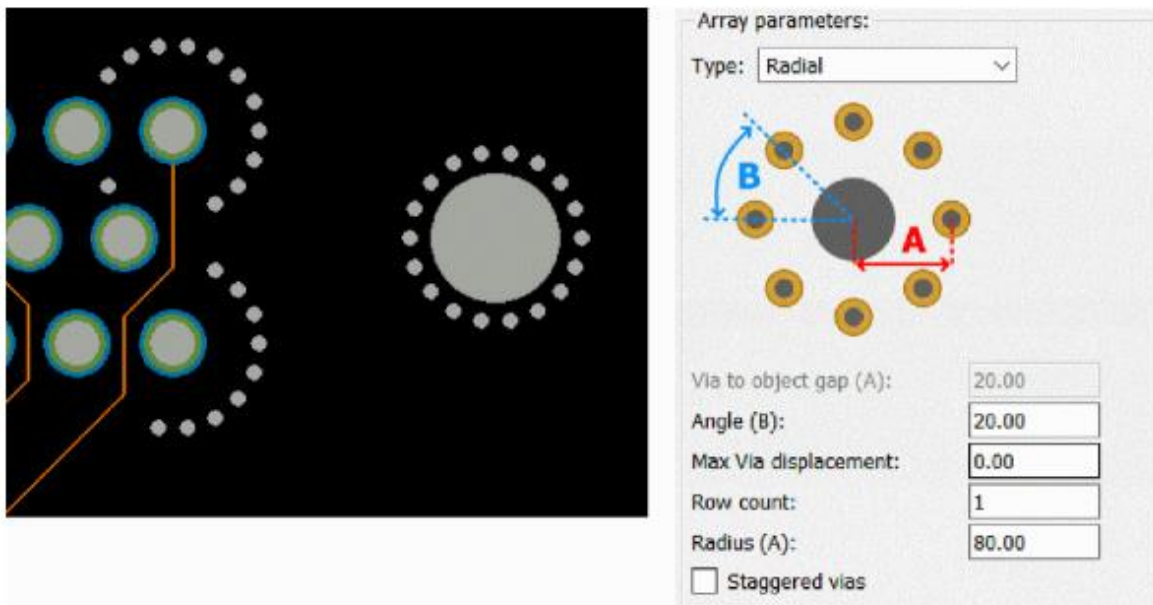
Surroundings

When the array type is set to Surrounding, the vias are placed along the external boundary of the shape according to the offset value you specify. The Via to Object gap determines the perpendicular distance between vias in the outermost ring and the shape boundary. The offset values defines the spacing between two adjacent vias in vertical and horizontal directions to determine the number of vias.



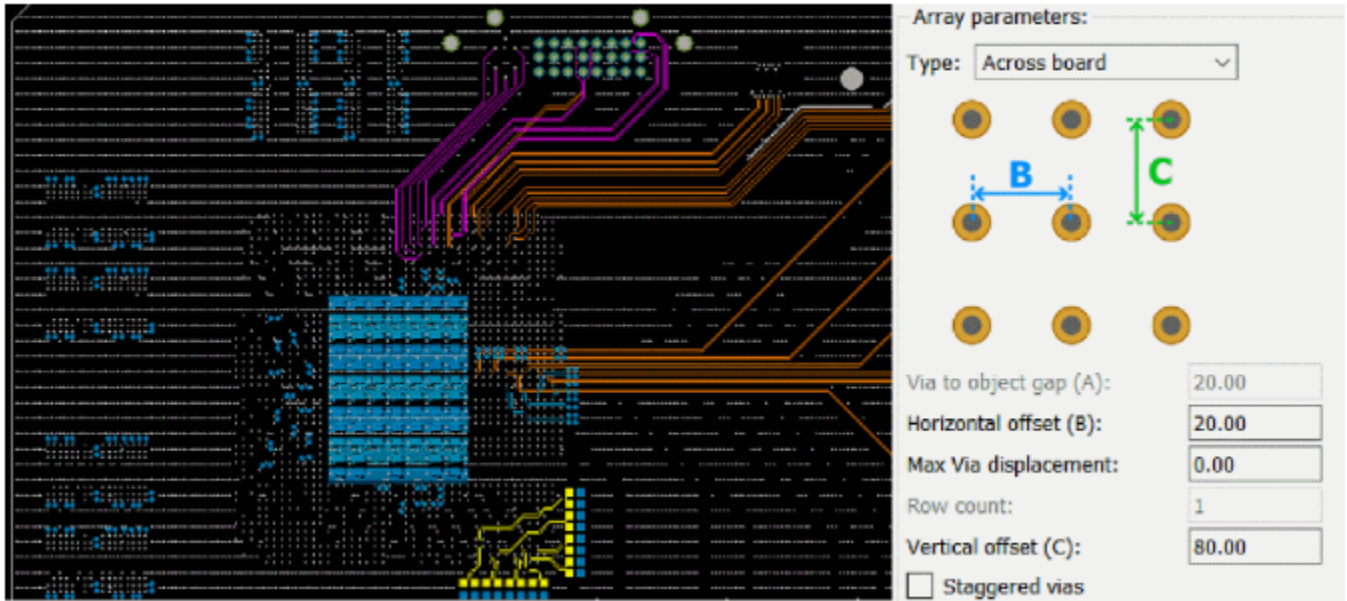
Radial

A radial via array places the vias or structures in one or more rings around a pin or via. The Radius defines the distance from the centre of the object to the centre of a via on the via ring.



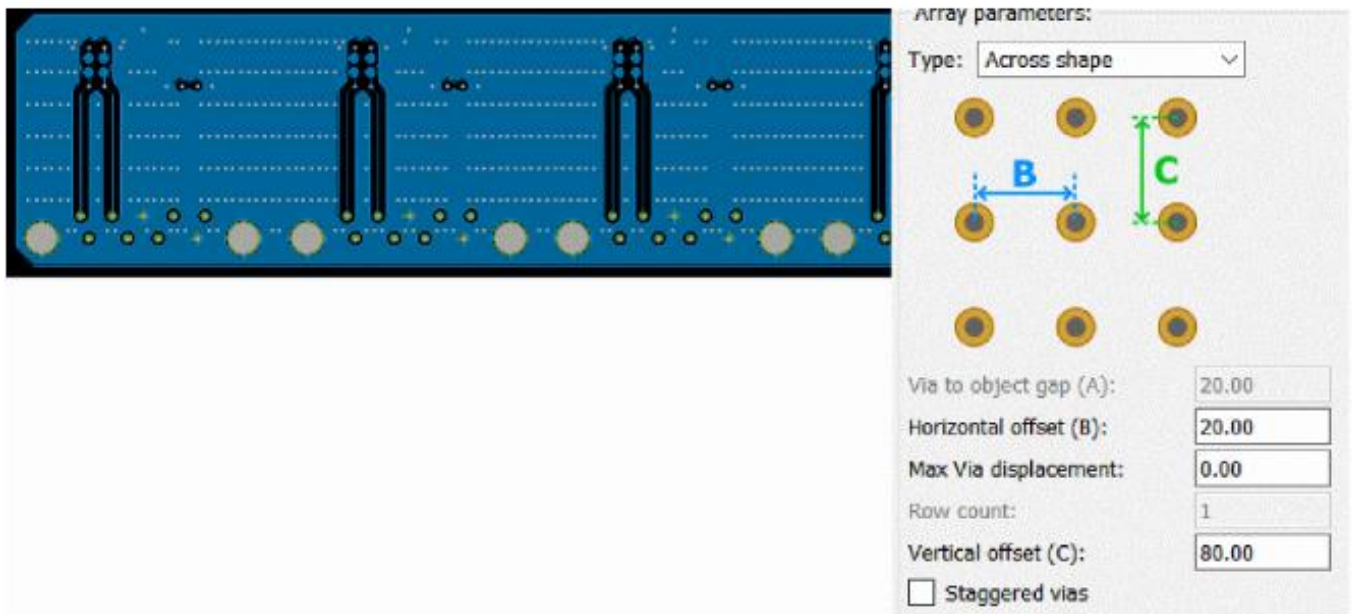
Across board

Set the array type and click anywhere in the canvas. Via arrays are placed inside the board outline.



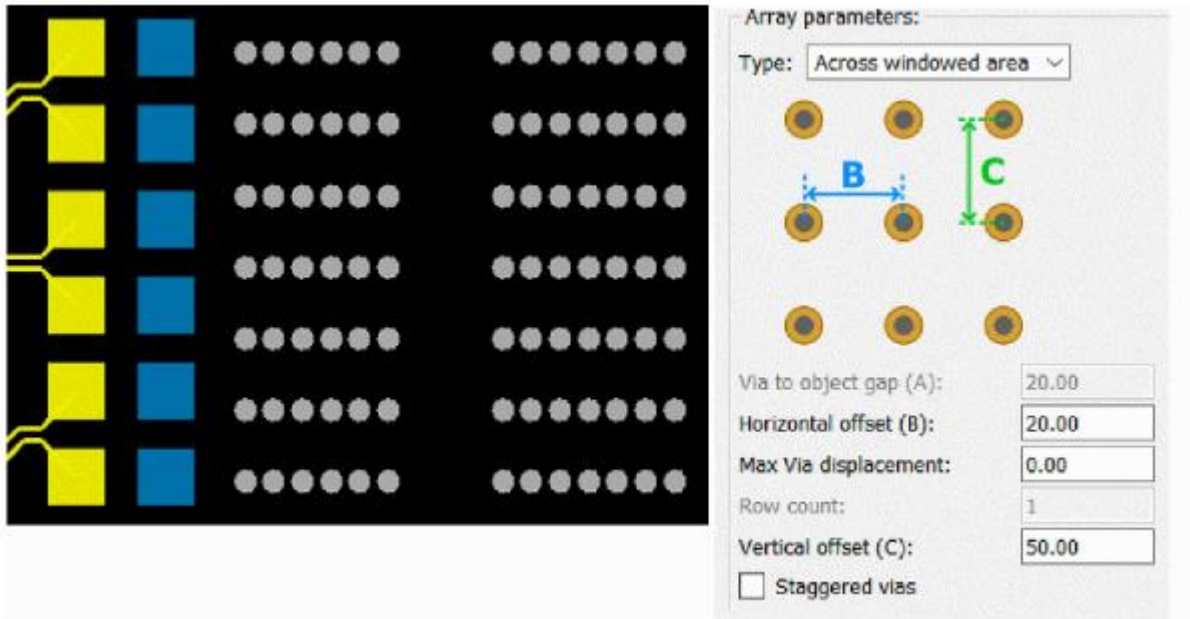
Across Shape

Select any shape and click anywhere in the design canvas to place the via arrays.



Across windowed area

Click anywhere in the design canvas, drag the mouse to create a window.



Updating and Deleting a Via Array

To update or delete a via array, enable update or delete mode in the General Options. Change the settings and select the object or array. You can see the preview of the updated via array if the Enable Preview option is checked. Clicking in the design canvas confirms the update or delete action. You can see the number of vias added or deleted in the command window.

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.