

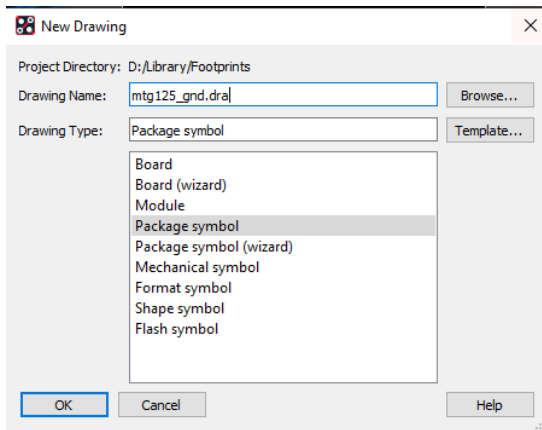


## Introduction

Many designs today require a good ground connection. Designers often use a “ring” of vias to surround the fixing hole to ensure this ground connection is made. This technical note describes how to produce this using the Allegro / OrCAD PCB Tools.

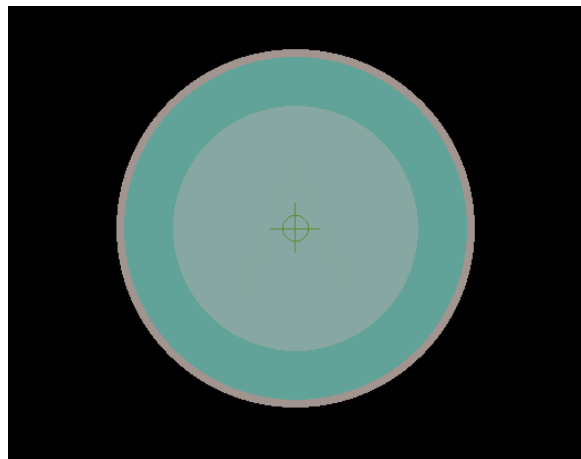
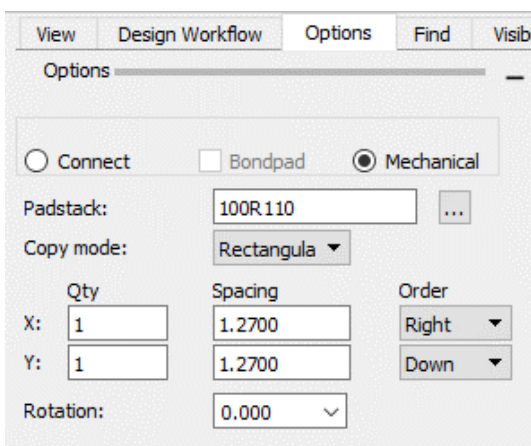
## Fixing Holes.

Start PCB Editor and use File > New. The following GUI will appear. You have the choice of whether to create a



package symbol or a mechanical symbol. If you want to connect the fixing hole to a ground net then you would need to include this part in the schematic so that it becomes a net-listed component and you then get a legal connection to the ground plane. For this option you need a Package Symbol. If you do not want the connections then you can use a Mechanical Symbol. For this example we are going to use a Package Symbol. Complete the choices and click OK.

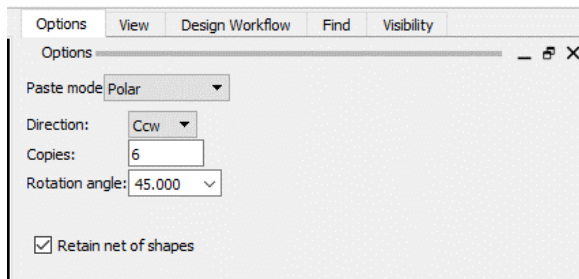
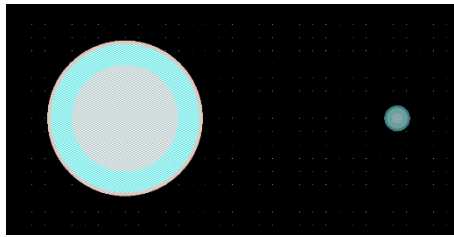
Setup the parameters for origin, colours, design size etc as you would normally. The first step would be to add the fixing hole at the origin point (0,0). Use Layout > Pins, browse to your required padstack and add the pin at the origin point. If you only want to connect the ring of vias to the ground plane, make the centre hole a Mechanical Hole. You can then clear this from any Copper areas.



The next step is to add the vias. If required you will need to use Constraint Manager to define your default via. This can be done through Setup > Constraints > Constraint Manager, Physical Tab, Physical Constraint Set > All Layers, define the vias under the DEFAULT PCSet. Click on VIAS to define your default ones. If you always use the default VIA then you do not need to complete this step. To add the via use the Layout > Connections

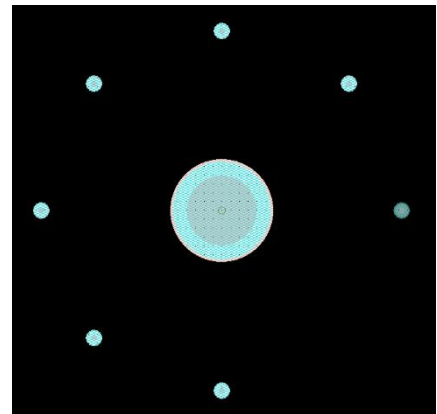


Pick a location for the via and double click. For this example, the via was placed at a location 5.08mm to the right of the fixing hole. If you want to use specific co-ordinates you can start the command then use x 5.08 0 followed by enter at the command line, then right click > Add Via. The result should be as follows:-

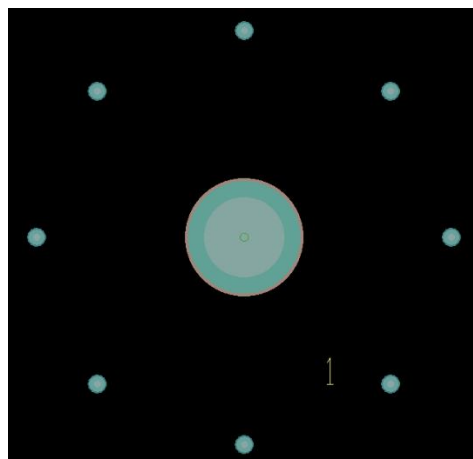


Next, we want to copy the via using the Copy command to generate a Polar Array. Use Edit > Copy, left click the via (to select) then in the options menu change the paste mode from Rectangular to Polar, then define the Direction, Copies and Angle. For this example the settings are as shown.

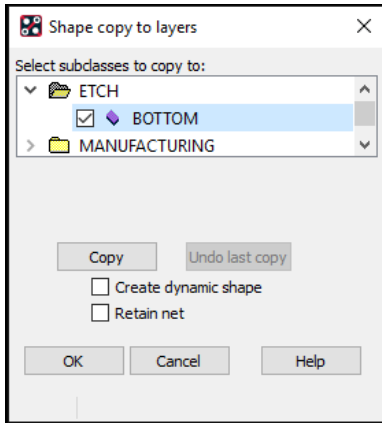
The Command line is now asking for you to select the destination of the polar origin. Left click the centre fixing hole or type x 0,0. The vias are copied around the fixing hole using the options supplied. The symbol now looks like the example shown.



Finally we need to add a Connect Pin that gives us the ability to join the via array to the ground plane using a schematic symbol. Use Layout Pins ensuring that the via padstack is selected and the pin is a Connect pin. Also ensure that the copy mode is Polar. Select the polar origin point (0,0) with the Left click. When the via is attached to your mouse type "polar" at the command line. A GUI appears asking for you to enter the radius, for this example enter 5.08, you are then prompted for the angle in degrees, enter 315 followed by enter. The pin is added at the location specified completing the polar array. The resulting array is shown below.

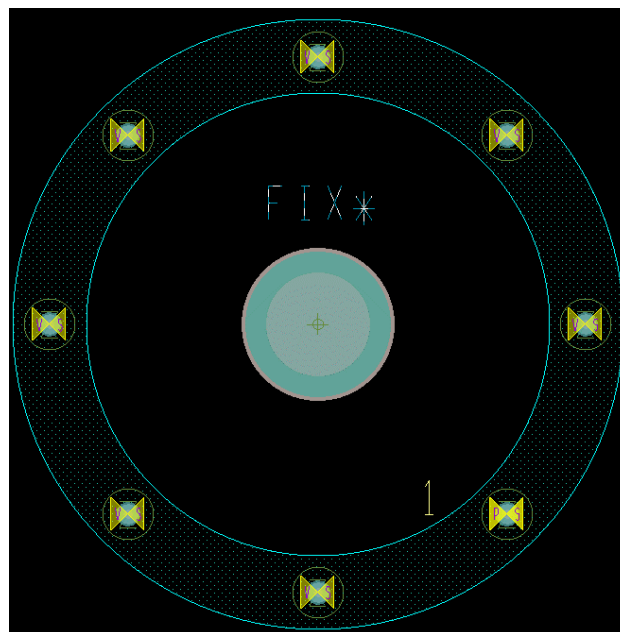


We now want to add a copper area to join the vias together. We can use Shape > Circular, set the class / subclass to Etch / Top, then pick the origin point or type x 0,0 at the command line, the diameter of the circle can be defined by either moving the mouse to the required diameter or typing the co-ordinates at the command line, This example used the command line method of ix 5.8mm (which is an incremental co-ordinate of 5.8mm from the selected origin point. Next (if required) we need to add a circular void to clear the centre fixing hole. Use Shape > Manual Void > Circular, Left click the shape to select it then type x 0,0 to pick the origin followed by ix 4.4 for the next selection. Then use Right click > Done to complete the void. If you prefer you can use the Left click to select the start and finish points of the outer and inner circles.



If you want to copy the shape to other layers like the bottom or soldermask layers, you can use Shape > Select Shape or Void, Left click the Top Copper Shape then Right click Copy to Layers; select which layers you want to copy the shape to. For this example Etch/Bottom and Package Geometry / Soldermask\_Top and Bottom have been selected, then press Copy and the Shape is copied to the three layers.

Finally use Layout > Labels > RefDes to define a Ref Des location like FIX\*. The completed symbol should look like the example below. Any DRC errors shown in the symbol will be removed once the netlist is imported and the symbol is placed into the Board.



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.