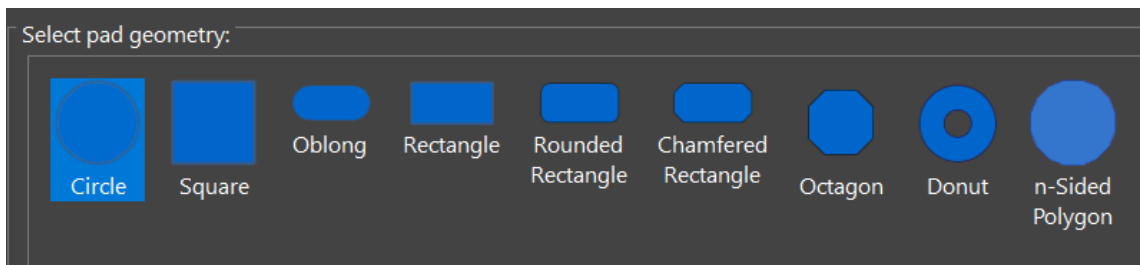




## Using Flash and Shape Symbols in PCB Footprints

### Using Flash and Shape Symbols in PCB Footprints

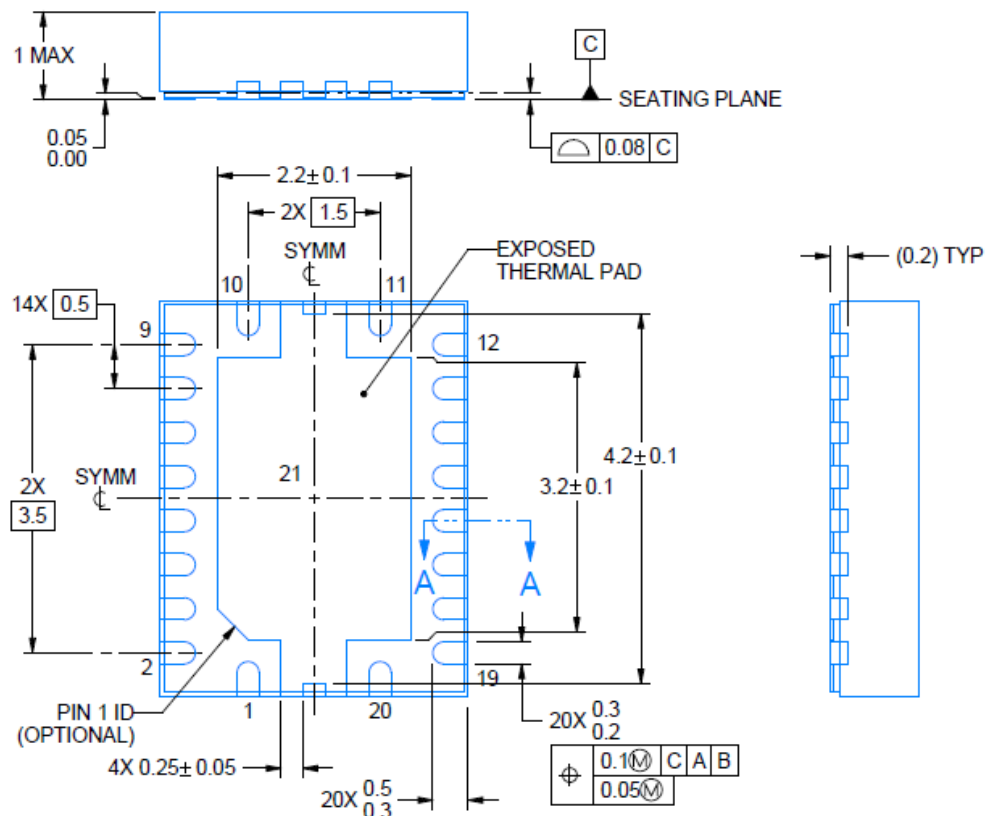
This app note will go through the process of using shape and flash symbols in a PCB Footprint. The main reason these are used is because many modern PCB Footprints require non default style pad shapes for copper, soldermask and pastemask openings. Cadence Padstack Editor has been greatly enhanced to include a variety of different pad shapes (shown below) but there are still many instances where more complex style shapes are required.

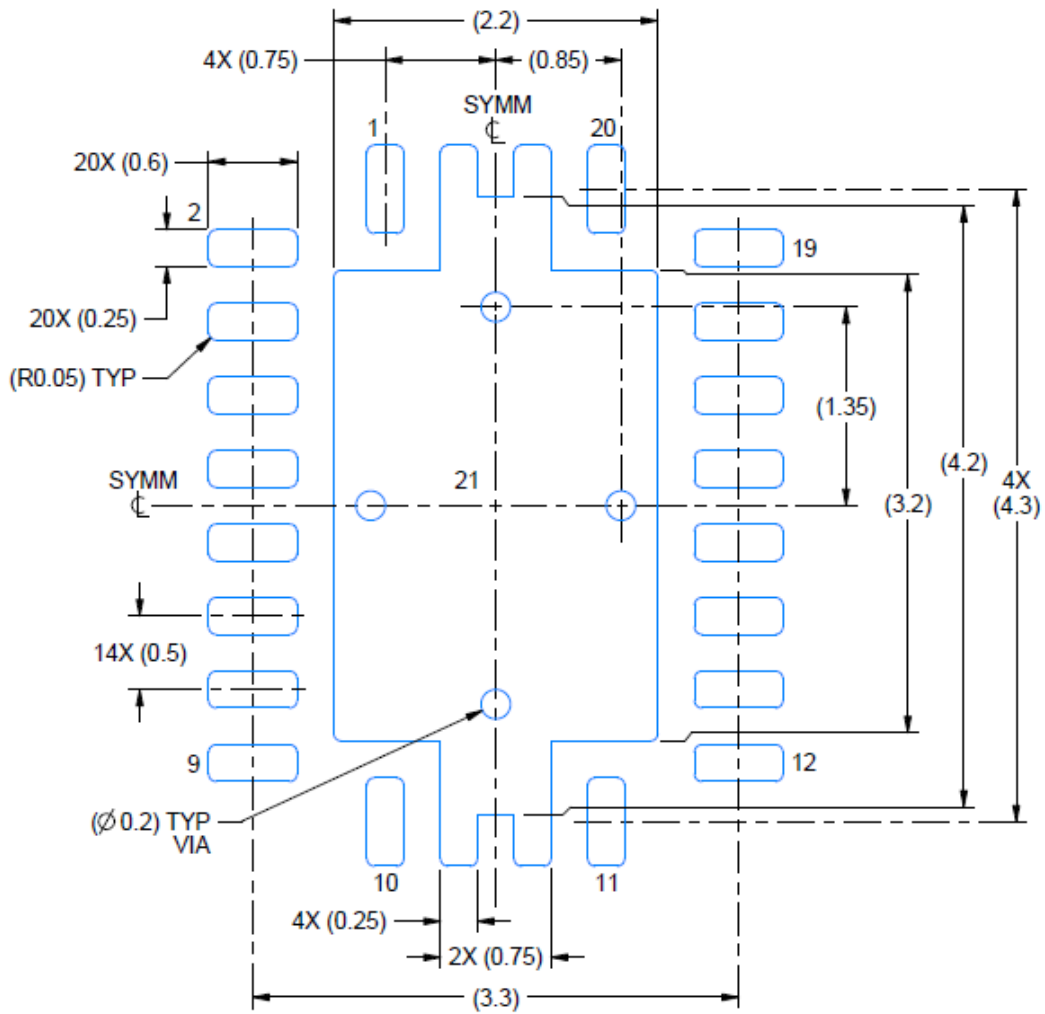


This app note will focus on the flash style symbols and also how to resolve any issues you may have using either shape or flash symbols in the Cadence PCB tools.

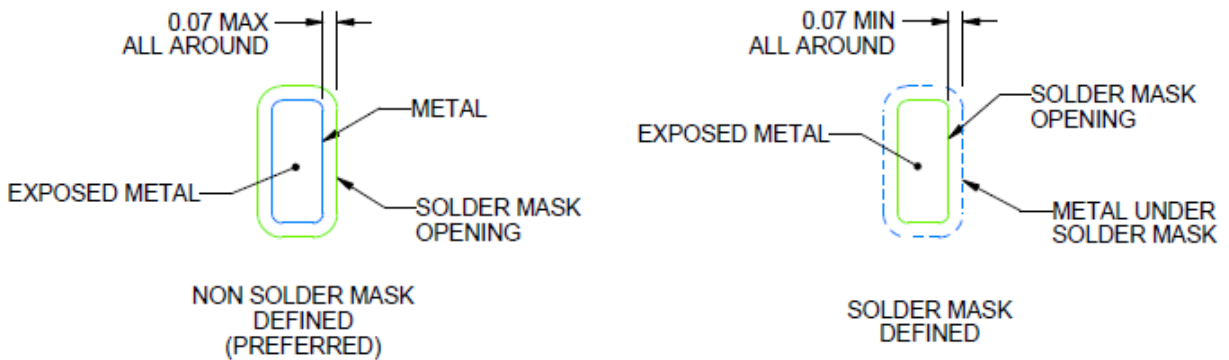
### Example Texas Instruments TCAN4550.

Below is a screenshot of the datasheet for the Texas Instrument part TCAN4550. It has different definitions required for copper, mask and paste openings on the PCB Footprint.

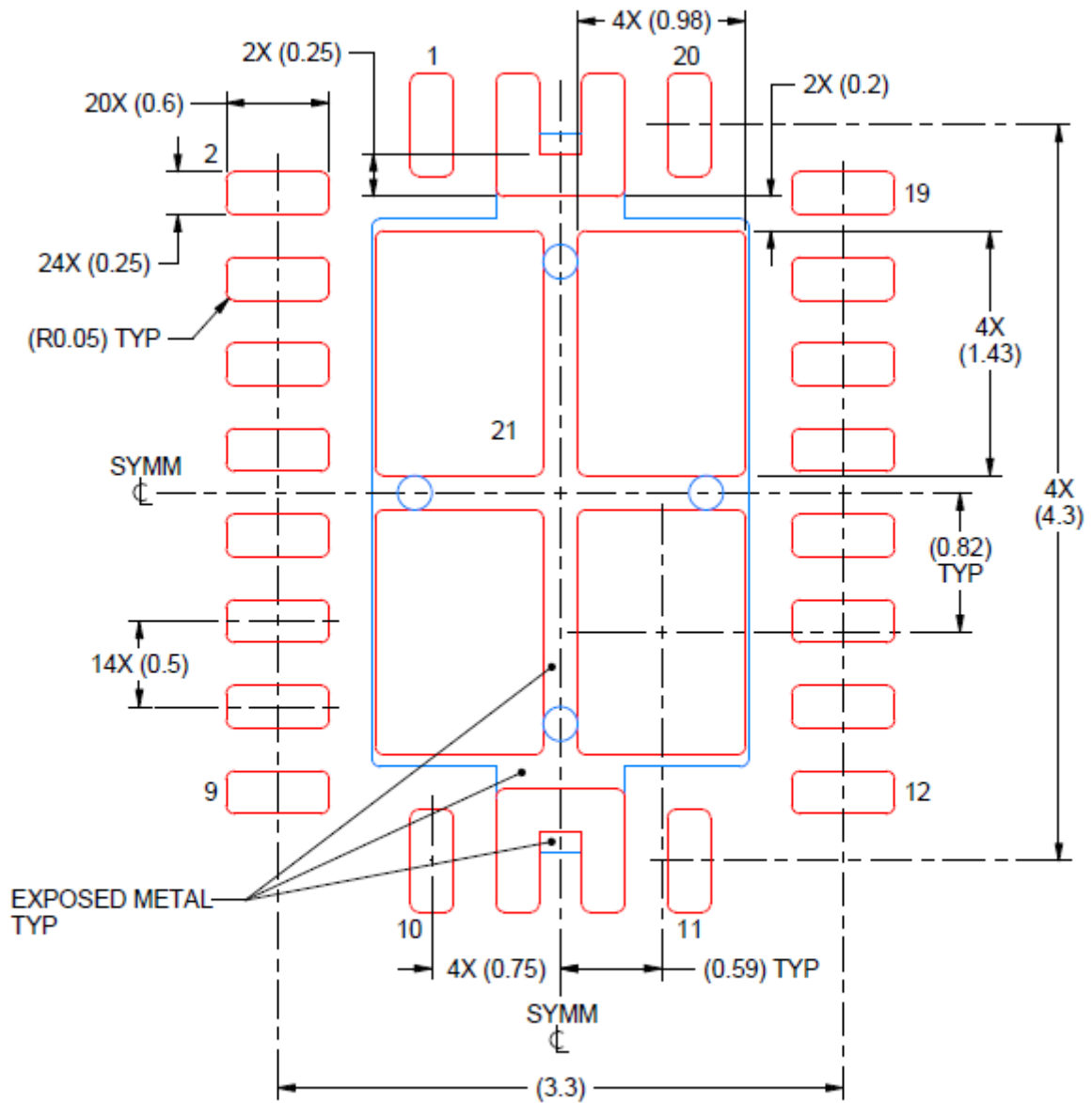




LAND PATTERN EXAMPLE  
EXPOSED METAL SHOWN  
SCALE:18X



SOLDER MASK DETAILS



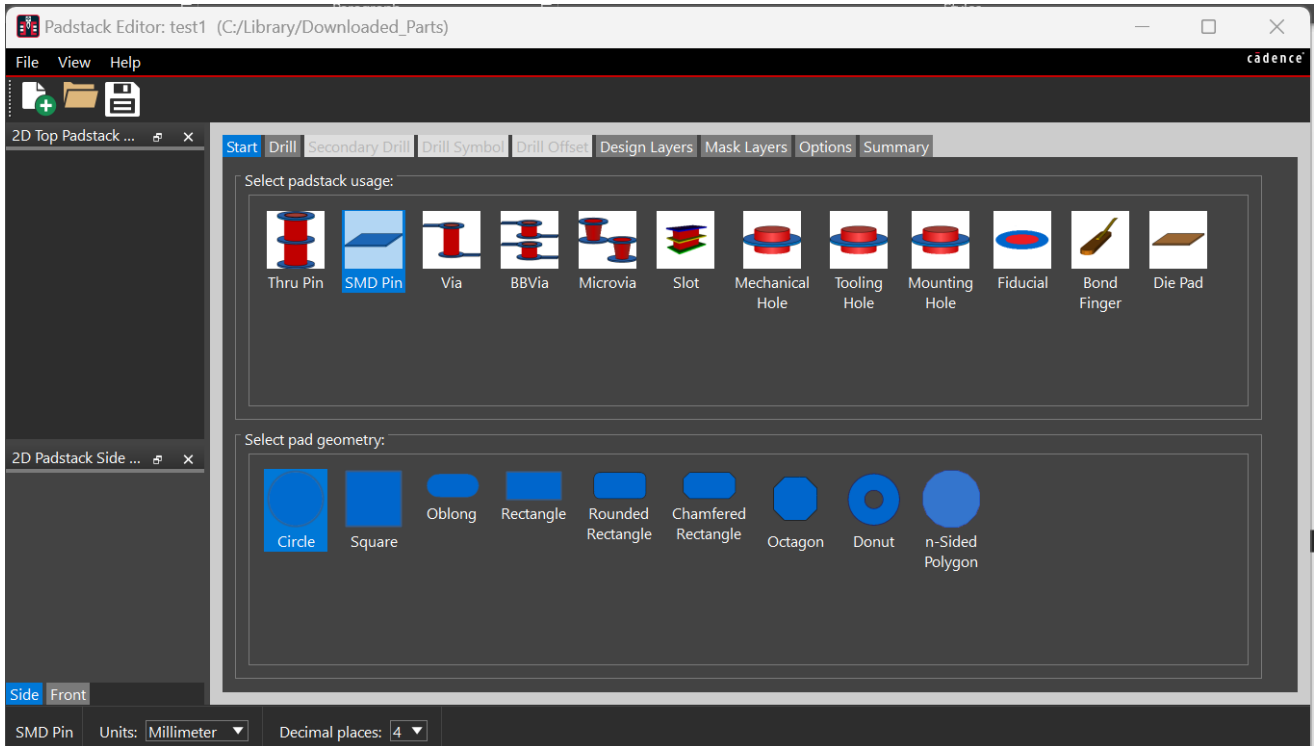
SOLDER PASTE EXAMPLE  
 BASED ON 0.125 mm THICK STENCIL

EXPOSED PAD 21  
 80% PRINTED SOLDER COVERAGE BY AREA UNDER PACKAGE  
 SCALE:20X

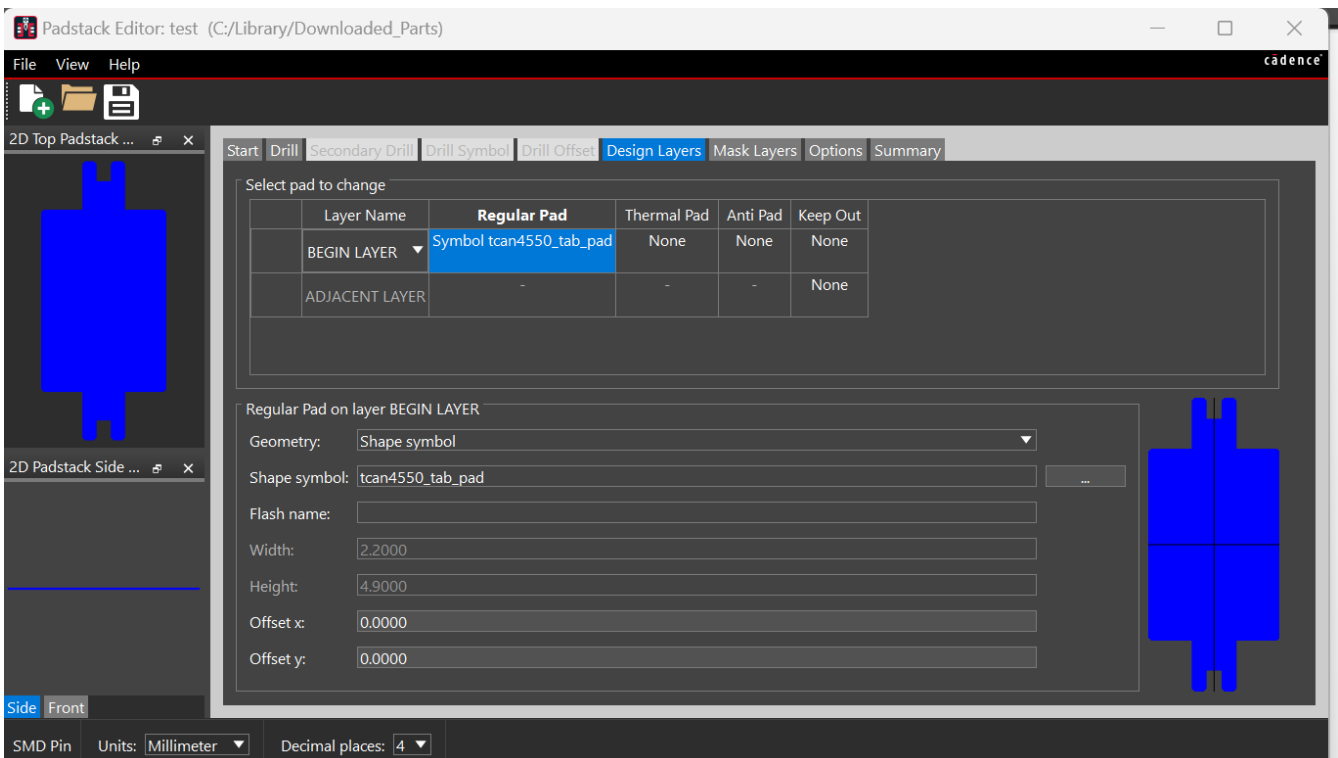
This app note is not going to go through the process of how to generate the shape and flash symbols. If you need assistance on those steps please refer to this app note [How to Create an alternative shaped pad/pin \(parallel-systems.co.uk\)](http://www.parallel-systems.co.uk) which describes that process in detail. The key difference to remember is that a shape symbol can only contain one static shape, a flash symbol can contain multiple static shapes. Both MUST be drawn on the ETCH/Top subclass and should be saved with their origins in the centre of the shape which will coincide with the padstack origin point.

## Using Flash and Shape Symbols in PCB Footprints

Once you have your shape and flash symbols defined using PCB Editor, launch Padstack Editor from the windows start menu found under the Cadence PCB Utilities VXX.XX menu. Use File – New and specify a name and padstack usage. In this example it is a SMD Pin. The units have also been defined as Millimeters.

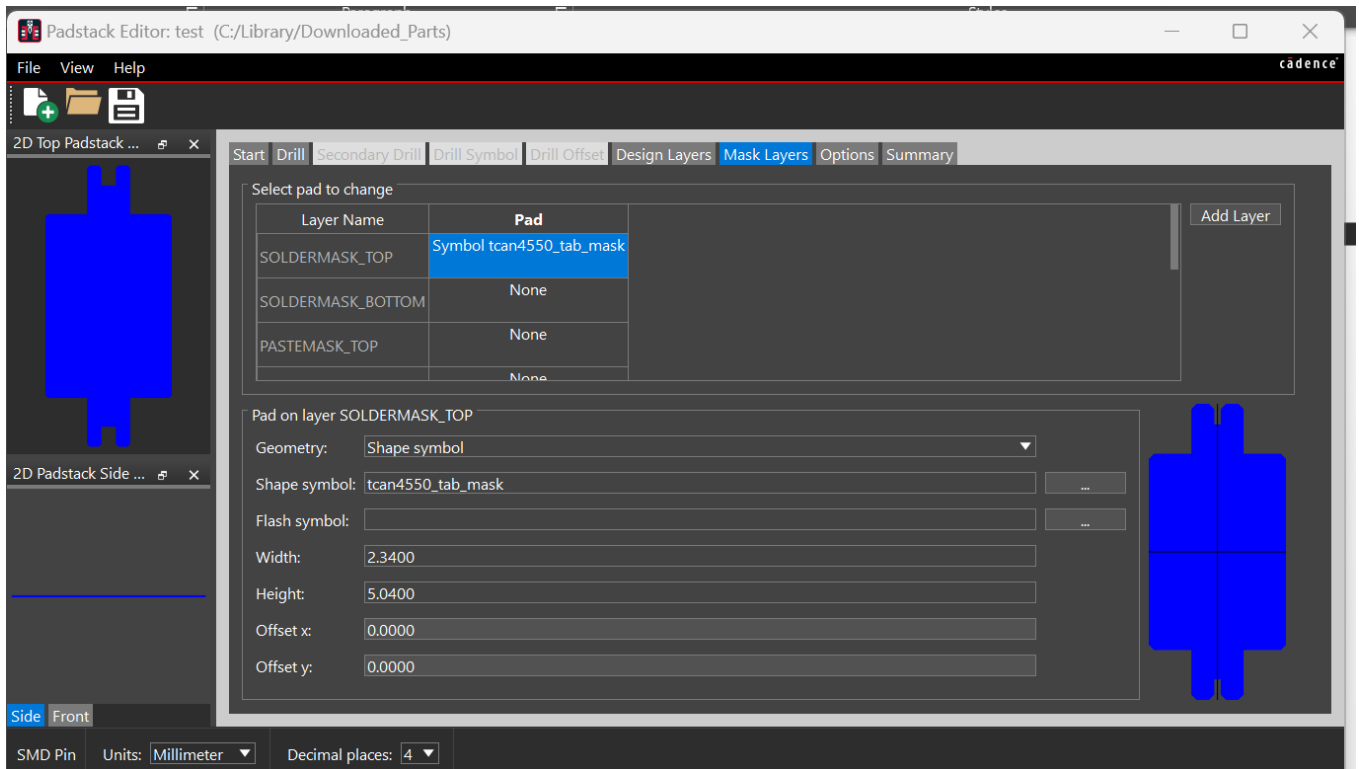


We don't need to define any drill settings (SMD Pin type) so move straight to the Design Layers tab. Specify the pad geometry as Shape Symbol and browse for the shape symbol you have generated for the copper tab pad. The browse button will show entries from the psmath location so make sure you save the filename.dra and filename.ssm or filename.fsm in this location.

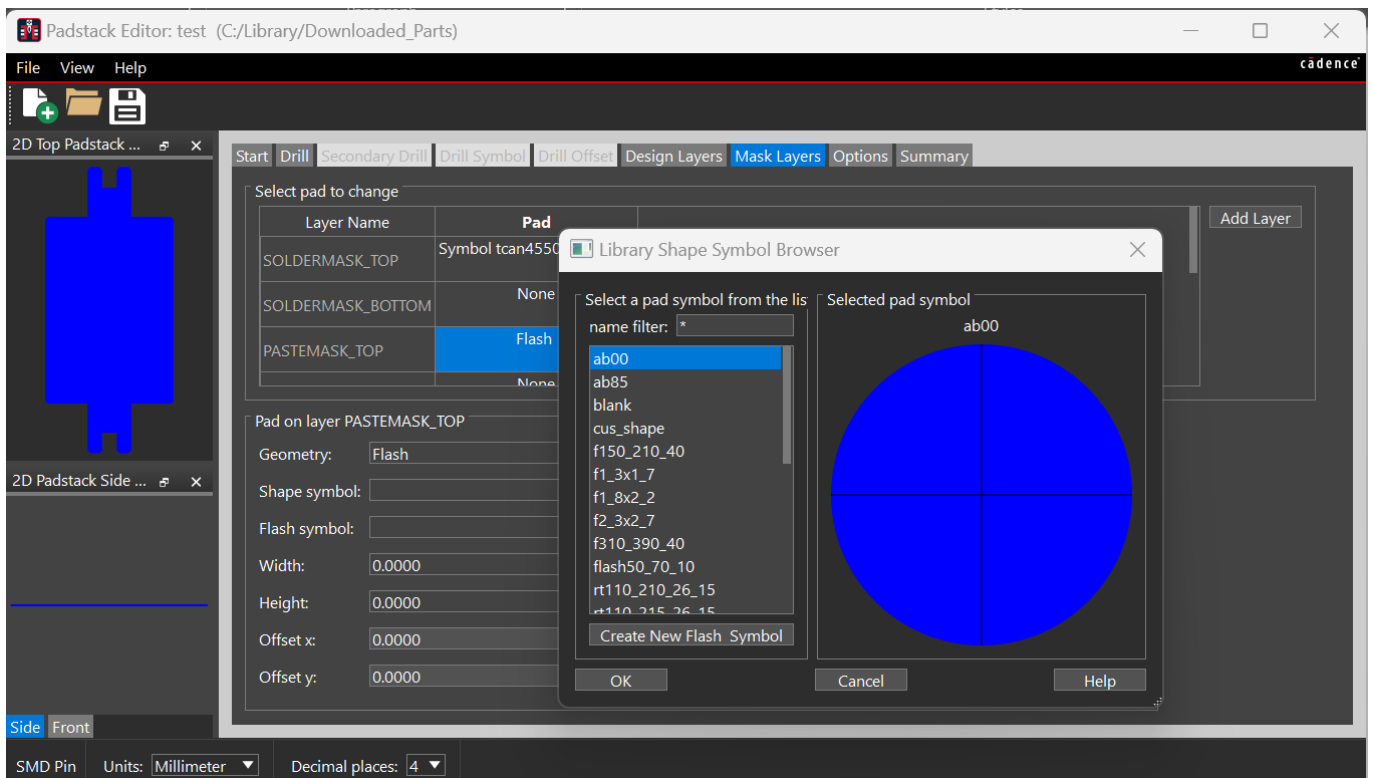


## Using Flash and Shape Symbols in PCB Footprints

Open the Mask Layers tab and repeat the steps for the copper pad but this time use the shape symbol generated for the soldermask opening.

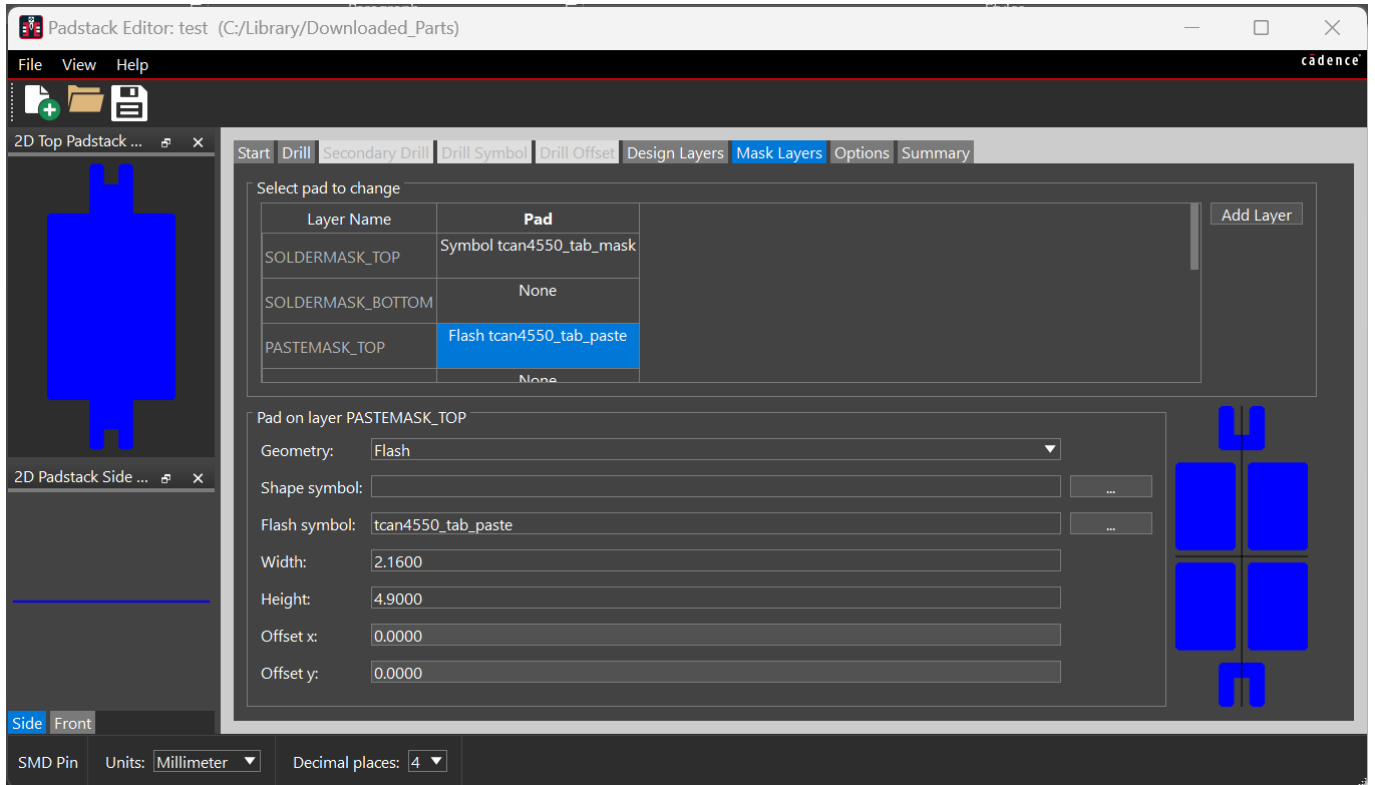


Now for the Pastemask opening we want to use a flash symbol. Select the pastemask\_top entry and change the geometry to Flash. You can now browse for an existing Flash symbol or use the "Create New Flash Symbol" button to launch PCB Editor and generate one.

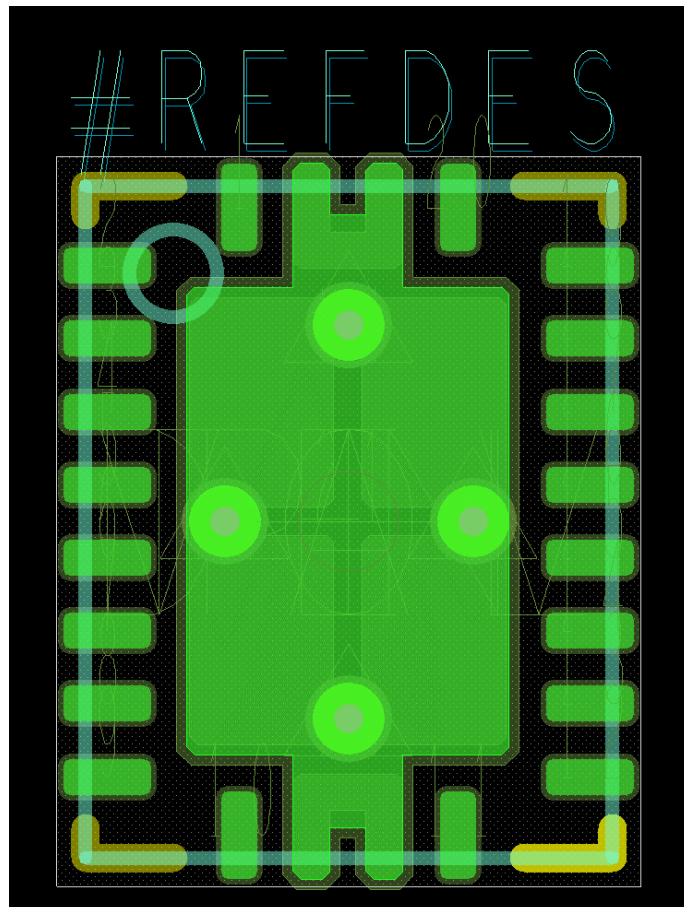


## Using Flash and Shape Symbols in PCB Footprints

Once saved you can select the flash symbol and add it. The biggest advantage of using a flash symbol over a shape symbol is the ability to add multiple shapes to one symbol.



Once complete, save the pad and place it in your footprint. The completed footprint is below.

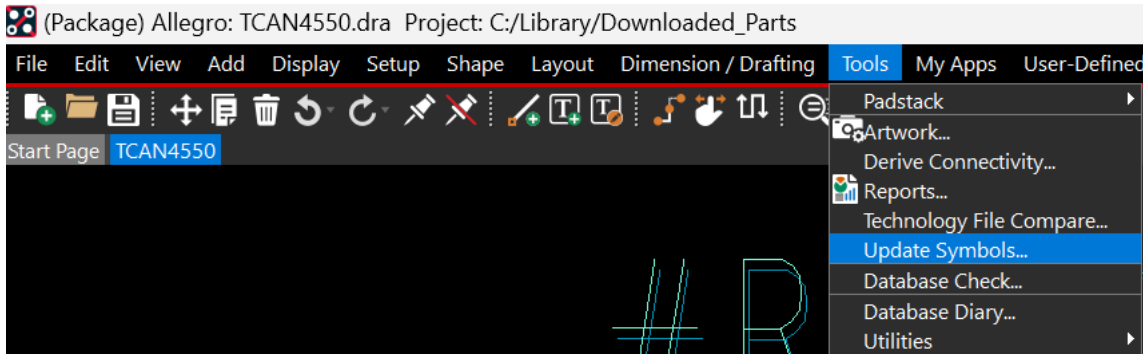


### Trouble shooting issues.

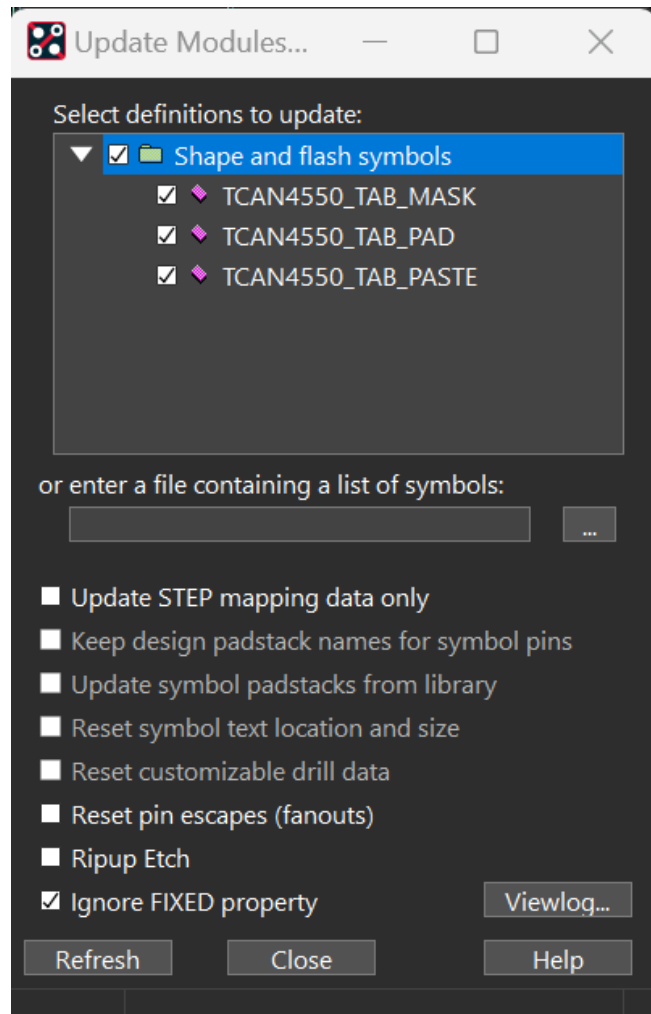
The key part to remember about using shape and flash symbols as part of a PCB Footprint is that whenever any type of PCB Footprint is used in a PCB Editor board file any symbols (package, mechanical, shape, flash or format) as well as padstacks are cached in the design. This means that if you need to make any changes to those items, they will need to be refreshed in the board file to show the updates. This are different methods for this depending on whether you are in a Package symbol or a board file.

### Package Symbol

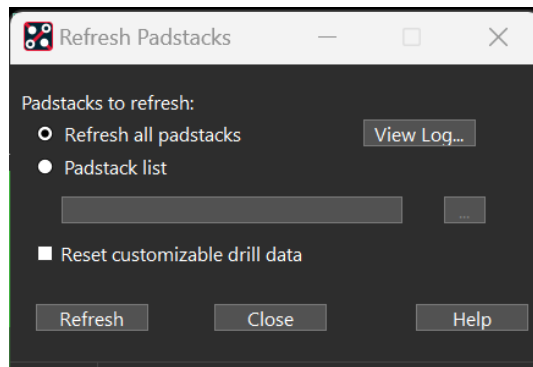
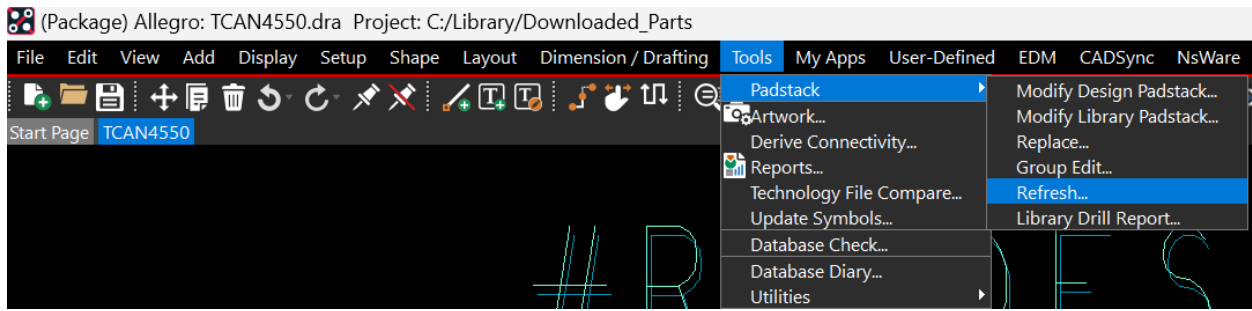
Use Tools – Update Symbols



Select any shape or flash symbols that you want to refresh / update. It is also recommended that you enable Ignore Fixed property then click on Refresh to update the symbols accordingly.

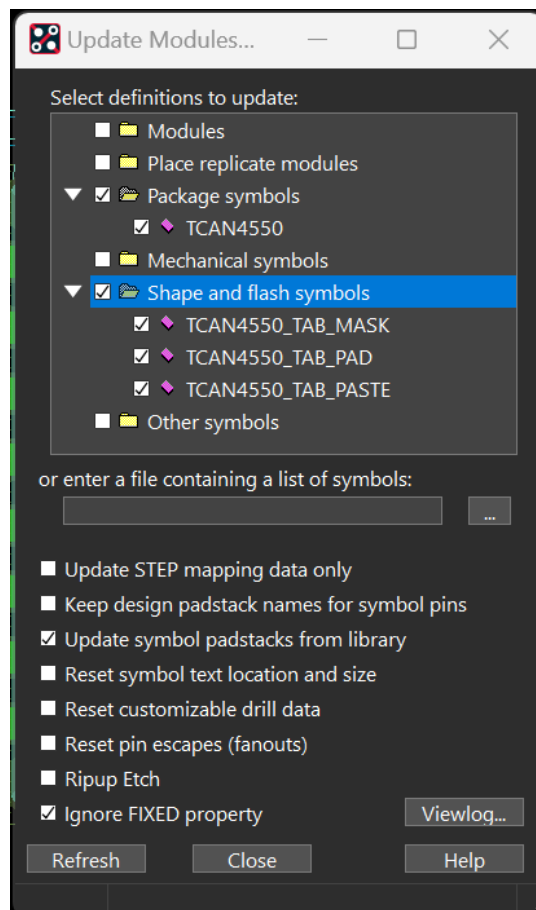


You may also need to refresh any changes to the Padstack definition. This can be done from Tools – Padstack – Refresh.



### In a Board File

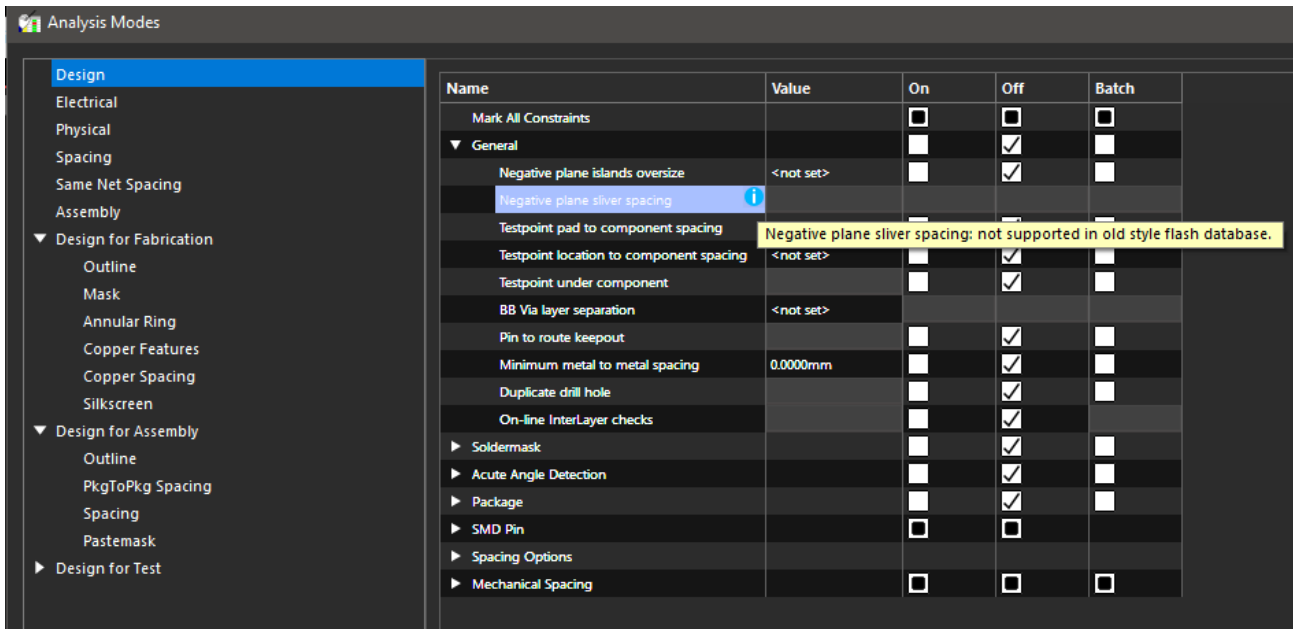
The steps are similar to a Package Symbol but in this case use Place – Update Symbols, enable the Package Symbol you require as well as any associated Shape and Flash Symbols. You can also enable Update symbol padstacks from library and Ignore FIXED property. Click on Refresh to update the required items.





### When an Update Symbols doesn't show the required results

There are some instances where you may have an old version board file (usually designed in pre-14.0 versions) where the design is set to use Old Style Flash Symbols. The best and the easiest way to recognize any other "old style flash" boards is to open constraint modes, Setup – Constraint Modes – Design, then General and Look at the design options - Negative Plane Sliver Spacing, it will say "Negative plane sliver spacing not supported in old style flash databases".



If this is the case, you need to save the board file and close PCB Editor. Open a Windows DOS command prompt and browse to the location of the board file and run the flash\_convert utility over the board file.

Current versions of the Cadence SPB / OrCAD tools no longer rely on the Windows PATH Environment Variable to function so the flash\_convert utility will need the full path specified in the command prompt to run – it will be in the product installation folder, then tools\bin\flash\_convert. For example:-

```
C:\Demos\SPB174\New-DemoP\allegro>C:\Cadence\SPB_17.4\tools\bin\flash_convert.exe steve.brd

FILE: steve.brd
Performing a partial design check before saving.
Writing design to disk.
'steve.brd' saved to disk.

C:\Demos\SPB174\New-DemoP\allegro>
```

After running the flash\_convert utility, the Flash Symbols should behave as expected.

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.