

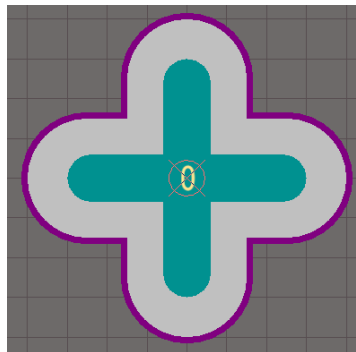
Footprint Creating Q & A

Question / Comment	Answer/Reply
<p>Is Class A/B/C the same as Class 1/2/3? Is it typical to specify this class on the fab drawing to the turnkey house? We don't typically communicate directly with the fab house</p>	<p>We stand corrected - it is Class 1, 2, and 3, where:</p> <p>Class 1 – General Electronic Products            Class 2 – Dedicated Services Electronic Products            Class 3 – High Reliability Electronic Products</p> <p>There are also Levels, as well which are defined as:</p> <p>Level A – General Design Complexity            Level B – Moderate Design Complexity            Level C – High Design Complexity</p> <p>Unfortunately, we didn't have time to touch on these. They are important, as well.</p> <p>Levels and Classes (along with Types) need to be discussed with the fabricator since they dictate the level of complexity, tolerances, and the pass/fail criteria for visual inspection. We also encourage you to seek the CID certification from the IPC. They go into more detail on this important subject.</p>
<p>The datasheet shows the bottom view of the device. The footprint needs to be constructed from the top view. The pin layout designed in the demo will not work. Needs to be mirrored.</p> <p>The pin designators are reversed the drawing is from the bottom and the PCB pattern is from the top</p>	<p>We stand corrected, though it would be nice if TI specified the view angle in the datasheet.</p> <p>This brings up several good points:</p> <ol style="list-style-type: none"> <li>1. This is why components should be reviewed by our peers.</li> <li>2. If one was to obtain a STEP model with the tab orientation, the STEP model upon placement would have shown that the footprint and tab were mirrored.</li> <li>3. When in doubt about the datasheet, contact the manufacturer.</li> </ol>
<p>Are the pad via library functions available for Altium 14?</p>	<p>The Pad Via library was introduced in Altium Designer version 15.</p>

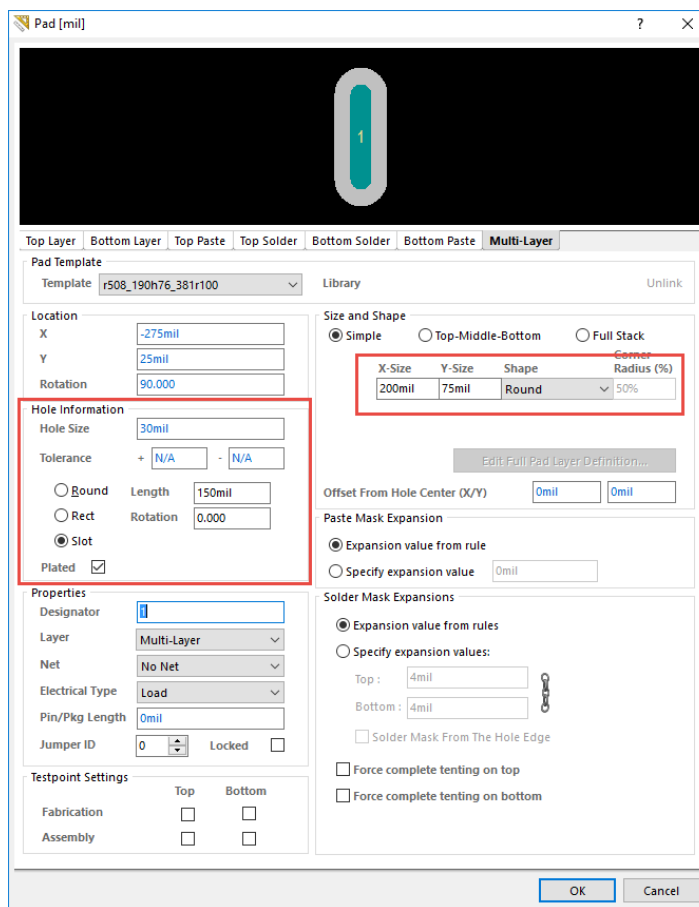
<p>What is the industry best practice for holes that do not require electrical connection (i.e., mounting features on connectors, holes for mounting the PCB without requiring electrical connection)? I've seen designs where these holes are NPTH (Non-Plated Through holes) and I've seen designs where they are PTH.</p> <p>I had heard once that NPTH are more difficult to manufacture. Not sure if that's true. I prefer not to have floating copper on the board.</p>	<p>Plating is primarily used for electrical and thermal connectivity, and it usually serves both the purpose of grounding the component and providing a path for thermal dissipation. If one is going to plate the hole, it is best that the copper be tied to ground. Otherwise, there is the possibility that it could become an accidental antenna.</p> <p>As for cost, it is easier to plate holes than to mask them since the board is plated "en masse" and not individually.</p>
<p>I'm assuming there is NOT a standard for mechanical layer usage, but if there is something recommended where would I find that information</p>	<p>You are correct – there is no standard for mechanical layers. If you are looking for a comprehensive list, check out this posting on Altium's forums.</p> <p><a href="https://forum.live.altium.com/posts/75191">https://forum.live.altium.com/posts/75191</a></p> <p>If you are going to search on forums, it is recommended that you type +Mechanical +Layers +Darren +Moore. Darren was instrumental in leading a number of forums posts on this topic.</p> <p>There are far more layers than what you will need. Use layers that make sense to you and your organization.</p>
<p>Where is the "convert special string" option?</p>	<p>Altium has available special strings in both the schematic and PCB editors. In the schematic editor, they start with an equal sign (=); in the PCB editor, they start with a period (.). By placing these special strings, Altium can provide information such as date, time, designator, etc. When one uses these special strings, the name of the special string is visible. To convert the special string to its value in the PCB editor:</p> <p>View Configurations (shortcut "L" key), View Options tab. In the region titled "Display Options", check the box for "Convert Special Strings."</p>
<p>For the odd shaped pad that is made using a polygon, Will the entire polygon be plated when the PCB is manufactured?</p>	<p>Regardless of the copper primitives used, as long as there is copper on the board, it will be present in the fabrication files (Gerbers, ODB++, IPC-2581). However, a word of caution when it comes to the solder mask and solder paste. One must enable these features if an odd shaped polygon pours (which Altium will convert to a copper region) is being used in a footprint.</p>

What about odd shaped through hole pads? For example, cross shaped through hole pad

You can do a slot in Altium Designer:



This is achieved by using the slot features in the pad properties:



In this example, 2 pads were used with the same designator; one was turned 90 degrees.