1. Altium Designer Tips & Tricks – TIP #08: PTH and NPTH pad definition

In the PCB designing, there are two types of pads: PTH (Plated Through Hole) and NPTH (Non Plated Through Hole). The first type is most often used for soldering and mounting the THT components. Whereas NPTH pad is often named as a mounting hole and fulfills that functionality.
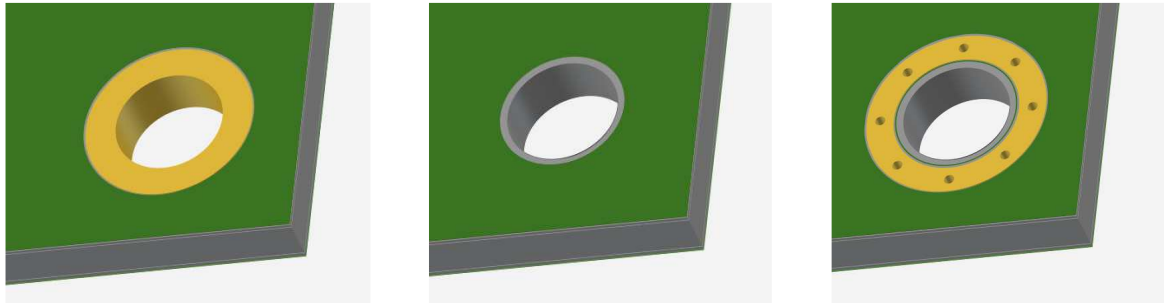

Fig. 1. Three different types of mounting pads: a) PTH pad, b) NPTH hole, c) NPTH hole with guarded ring

For proper NPTH pad and NPTH configuration use recommended values from Tab. 1. and also check an example of PTH/NPTH hole defined on Fig. 2.

	PTH pad	NPTH hole
Layer	Multi-Layer	Multi-Layer
Hole shape	Round	Round
Hole size	Your hole size	Your hole size
Plated option	Checked	Unchecked
Size and shape	Round, pad X/Y = size > hole size	Round, pad X/Y = 0
Solder mask expansion	Default (rule)	Manual solder mask pad size > hole size

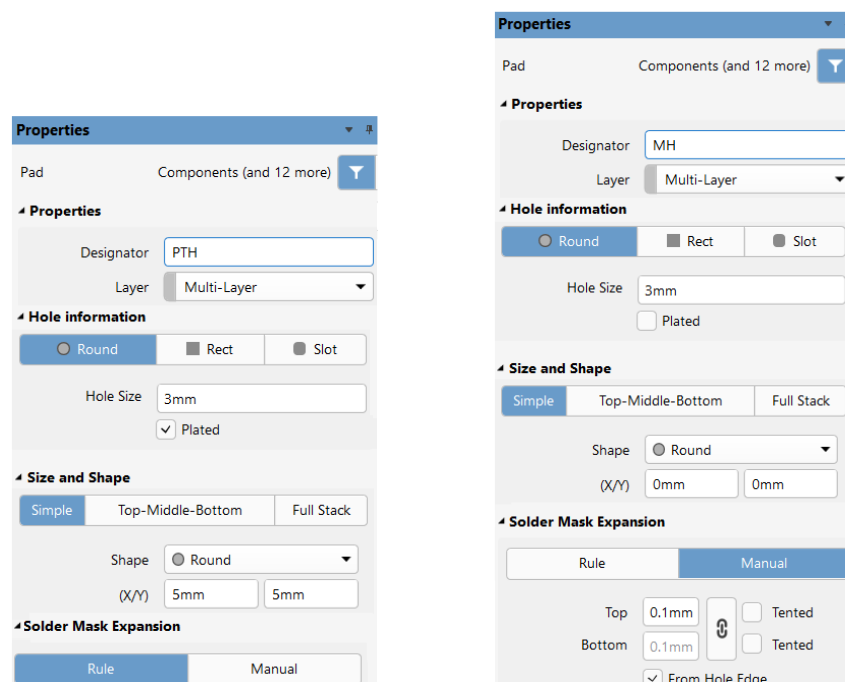Tab .1. Recommended settings for PTH and NPTH pads

Fig. 2. Example of: a) PTH pad definition (3mm hole and 5mm plated pad), b) NPTH hole definition (3mm hole)

TIP #1   For PTH definition always use option: "Plated=Checked" and define: "copper pad size > hole size".

TIP #2   For NPTH mounting hole always use option: "Plated=Unchecked" and define: "copper pad size = 0".

TIP #3   Add solder mask clearance around the NPTH hole. PCB manufacturers want to avoid route through a solder mask layer (dusty effect).

TIP #4   Avoid the two most definition errors for NPTH holes: checked option "Plated" (see Fig. 3.) and "copper pad size > 0".
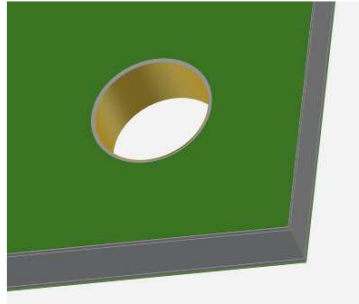

Fig. 3. One of the most common definition error of NPTH pad: checked option "Plated" for hole.