

How to Make PCB Mounting Holes in Altium Designer 2018

Before Designing

Are you grounding your board/chassis?

Avoid short circuits or loops by making sure your board and chassis are at the same potential. Also, choose between a plated or non-plated hole.

Are there any sensitive components near your mounting holes?

Components like ceramic capacitors can break quite easily due to the board flexing when mounted. It is best to keep them away from your mounting holes.

Step 1: Choose a Screw Size

There are many screw sizes to choose from and the one that is best for your design may not be the best for someone else's. There are many resources available online to learn about the different mounting hardware, but it is best to stick to the ANSI or ISO standard sizes to ensure proper mounting (Figure 1).

Screw Size	Hole size	
	ANSI (mils)	ISO (mm)
M2	86.6	2.2
M2.5	106	2.7
M3	126	3.2
M3.5	150	3.8
M4	169	4.3
M5	209	5.3
M6	252	6.4
M8	331	8.4
M10	409	10.4
#2	89	2.26
#4	116	2.95
#6	144	3.7
#8	170	4.3
#10	196	5
1/4"	257	6.5

* some example of screw hole size and hole sizes are approximate

Figure 1: Various standard hole sizes

Step 2: Add the Hole in Altium Designer

To make a mounting hole in Altium place pads (Figure 2) on the PCB corresponding to where you wish to place your mounting holes. After placing the holes use the properties panel to change the hole size to fit the mounting hardware and make the pad size smaller than the hole (Figure 3). Then uncheck “plated” if you wish to make a non-plated hole. (Figure 4)

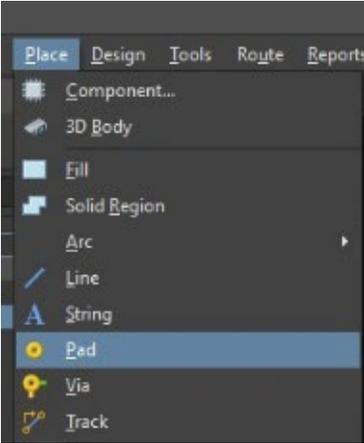


Figure 2: Pad place

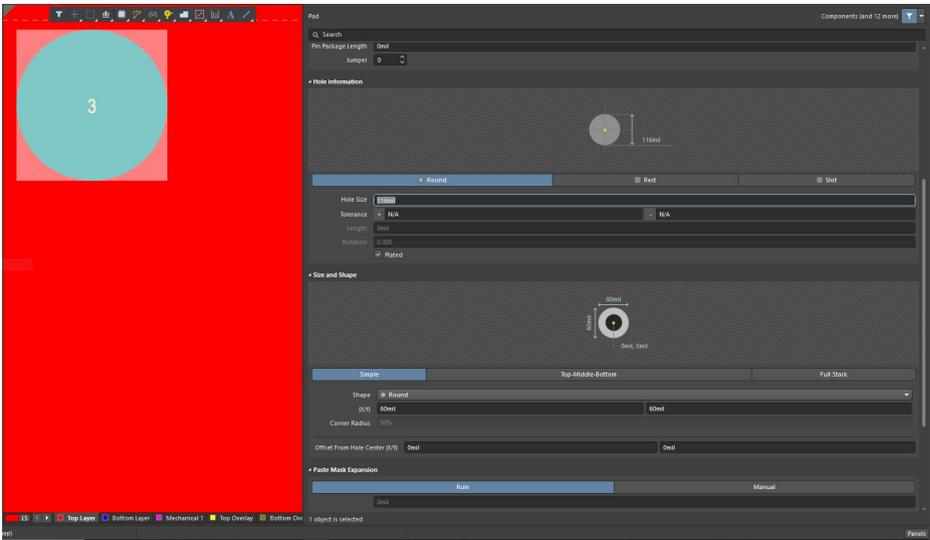


Figure 3: Properties panel for plated hole

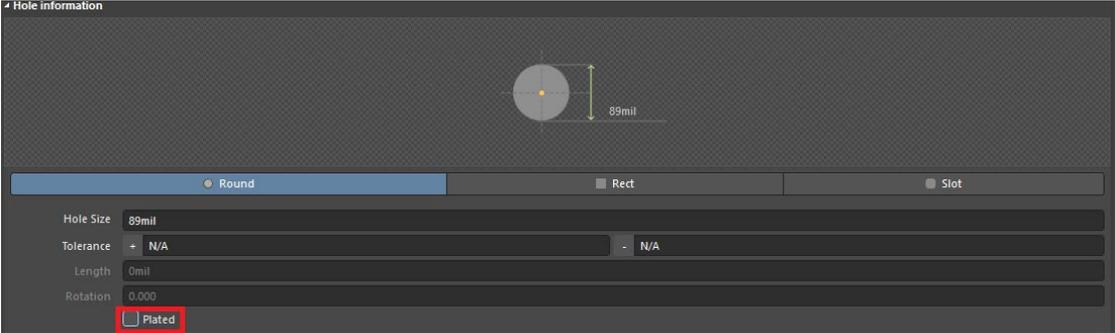


Figure 4: Properties panel non-plated hole