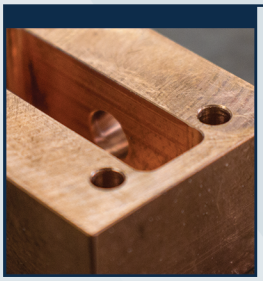
TOP WAYS TO SAVE ON CNC Machining

1 Internal Corners & Deep Pockets

Use an inside corner radius with a depth-to-tool diameter ratio of 3:1 or less and avoid varying internal corner radii. Design lengths up to 4x their depth.

2 Limit Tight Tolerances



Assign numerical values to critical features and surfaces, and keep all other features as standard tolerance: +/- 0.005". Define a single datum, like the intersection of two sides, as a baseline.

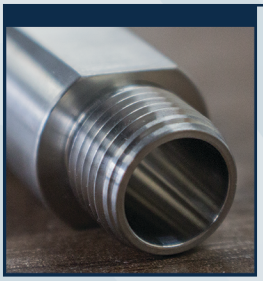
3 Expand Thin Walls

Walls should have a minimum width of 1/16" (1.5875mm). This is to avoid both chatter and the need to slow down machining speed.

4 Drill Sizes & Tapped Holes

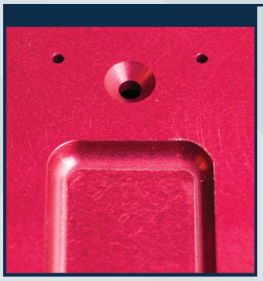
Utilize standard tap sizes such as 4-40 taps. Utilize standard fractional, number or letter drill sizes. Avoid very small holes and avoid threading more than 3x the hole's diameter.

5 Avoid Expensive Materials



Use a softer metal like aluminum 6061. If you need something harder, use a more machinable mild steel.

6 Avoid Multiple Finishes



When it comes to surface roughness specifications and other appearance-altering finishes, either leave the finish as standard or request a uniform finish.

7 Split Up Complex Parts

Especially useful for parts with deep pockets or multiple faces that require operations: design and machine as separate parts, then weld or bolt together.

8 Order Larger Quantities

Order multiple quantities of the same part (up to 10,000 parts for CNC) to reduce unit costs.

Xometry | Your one-stop shop for manufacturing on demand

Get an instant quote on your CNC parts at xometry.com today!

XOMETRY.COM/SUPPORT
SUPPORT@XOMETRY.COM
240-252-1138