The Tolerance On Your Drawing Is Not The Tolerance You Will Get

How to read a stamped or deep drawn drawing before it is quoted - which tolerances the process holds by default, which ones need a secondary operation, and what to mark up first.



DEEP DRAWING & METAL STAMPING
Zhuji, Zhejiang, China

BALFORD.NET
shawn@balford.net
2026-10-13

Almost every argument about a stamping quotation is really an argument about tolerances. It just starts later, when the first-article report lands.

The reason is mechanical, not commercial. On a stamped part, tolerances come from the station that made the feature, and different stations behave very differently.

Pierced and blanked features - holes in a flat, trimmed outlines, cut edges - come off a cutting station with no forming strain in them. In thin material they can hold a tight block tolerance without drama. Formed features cannot. Bending and drawing add springback, wall thinning and radius variation, and those three effects move the dimension after the tool has stopped moving.

Hole-to-hole position is a separate story again. It is set by the piloting and the strip layout, not by the tolerance block: if the tool pilots off one station and the drawing dimensions from another, the number in the title block is decorative.

The one that catches most programmes is wall thickness. On a drawn part, wall thickness varies with the draw ratio. It is a consequence of the process: material that was in the blank has to go somewhere when the wall gets taller. If a drawing asks for a uniform wall over the full height of a drawn housing, the honest answer is usually to form close and machine the wall, or to rewrite the requirement - not to promise it from the die.

A few habits make quotations much more useful:

- Separate cut features from formed features before you issue the drawing.
- Check the datum scheme. A part dimensioned entirely from a formed edge is slow to inspect and therefore more expensive to make than the geometry suggests.
- Watch stacked tolerances. Three features at plus/minus 0.05 mm can eat an assembly allowance between them before anyone has made a part.
- Mark two or three critical characteristics. Twenty unmarked tolerances produce a worse price than three marked ones, because unmarked tolerances have to be defended everywhere.

Where a feature genuinely needs more than forming can hold - a bore diameter, a seal face, a thread - the practical route is to form close and machine that feature, with its own inspection. Chasing it purely with the tool means a moving target at every sharpening, and the cost reappears later as sorting.

So when a quotation arrives, read what it says about tolerances. Which tolerances are proposed as standard for your material and thickness, which features are proposed for secondary machining and why, how the critical characteristics will be inspected and how often, how many tool stages are assumed and at what volume. A quotation without a tolerance discussion is a placeholder; the number only becomes real once both sides agree which tolerances govern the price.

The longer version, with the tolerance-by-process breakdown, is here: https://balford.net/metal-stamping-tolerance-standards-reading-a-drawing/

I am the founder of Balford, a deep drawing and metal stamping manufacturer in Zhuji, China. We deep draw and stamp metal housings on presses from 25 t to 350 t, to a maximum draw diameter of 250 mm, and quote from the drawing. The first thing we mark up on an incoming drawing is usually not a dimension - it is a tolerance that the process cannot hold.

SEND A DRAWING, GET A REAL ANSWER

Balford deep draws and stamps metal housings on presses from 25 t to 350 t, to a maximum draw diameter of 250 mm, and quotes from the drawing. **balford.net** · **rfq.balford.net** · shawn@balford.net