



Balford.

TECHNICAL WHITEPAPER · WP-02 · 2026

DRAWN HOUSING TOLERANCES

How to dimension a deep drawn housing so that it can be inspected, and what the process can and cannot hold. Written for the engineer writing the call-outs and the inspector who has to prove them.

CONTENTS

SPECIFY, MEASURE, VERIFY

01	WHY A DRAWN PART RESISTS DIMENSIONING	03
	The wall tapers, the part springs, and nothing is a cylinder	
02	THE DATUM SCHEME	04
	Three datums that stop two inspectors measuring two parts	
03	THE SEVEN CALL-OUTS	05
	What each one means, what it costs, what buys it tighter	
04	WHAT THE PROCESS DOES TO THE WALL	06
	Thinning, work hardening and what ironing changes	
05	SPRINGBACK	07
	Why the diameter is never exactly what the die says	
06	INSPECTION METHODS AND THEIR LIMITS	08
	What each instrument actually proves	
07	WRITING THE TOLERANCE BLOCK	09
	A worked set of call-outs for one housing	
08	FIRST ARTICLE AND PPAP	10
	Where the tolerance block meets the evidence	
09	DISPUTES AND HOW TO PREVENT THEM	11
	The four arguments that repeat, and the sentence that ends each	
10	THE TOLERANCE CHECKLIST	12
	What to settle before the drawing is released	

The one-line version

A drawn housing can hold almost any tolerance you want, provided you accept the operation that buys it.

Where the numbers come from

Ranges marked as typical are as-drawn results we plan with.
Standards are quoted as published.
No customer measurements appear here.

Standards referenced

ISO 2768-1, ISO 1101, ISO 5459, ISO 8015. State which edition governs the drawing.

CHAPTER 01

WHY A DRAWN PART RESISTS DIMENSIONING

A machined part is dimensioned against surfaces that exist. A drawn housing is dimensioned against surfaces that were formed by flow, that taper, that spring back, and that are measured on a part which is not a cylinder.

Four things that move

- **The wall tapers.** The diameter at the rim is not the diameter at the bottom, because the material was stretched over a punch and released.
- **The part springs back.** Elastic recovery after the punch withdraws changes the diameter by an amount that varies with material and wall.
- **The wall thins where it bends.** The thinning is not uniform around the circumference.
- **The rim is not level.** Earing and trim tolerance both leave the top edge uneven.

What that means for a drawing

A diameter is not a property of the part. It is a property of the part, at a height, measured from a datum, in a stated plane. Every one of those qualifiers has to be on the drawing, or the tolerance means something different to everyone who reads it.

The same applies to roundness: a wall that is round at the bottom can be out of round at the rim, because that is where the least constrained material ends up.

WHAT CHANGES THE MEASUREMENT, BEFORE ANY TOLERANCE IS APPLIED

VARIABLE	EFFECT ON THE READING	HOW IT IS CONTROLLED
Height of measurement	The largest single source of disagreement. A tapered wall moves the diameter by more than the tolerance.	State the height and the datum it is taken from.
Clamping and support	A thin wall distorts under a clamp. The reading depends on how the part was held.	State the support condition, or measure free-standing.
Temperature and relaxation	A part measured straight off the press reads differently from the same part the next day.	Measure after the part has cooled and stabilised.
Circumferential position	Wall thickness and roundness vary around the circumference, not just along the height.	Sample more than one position on the first article and record which.

The practical consequence

On a drawn housing, an unqualified diameter call-out is not a loose requirement. It is an unmeasurable one, and it will be argued about at first article. Qualifying it costs one line on the drawing.

CHAPTER 02

THE DATUM SCHEME

A housing is located before it is measured. Three datums do that job, and they are the same three whether the part is checked on a surface plate or on a CMM.

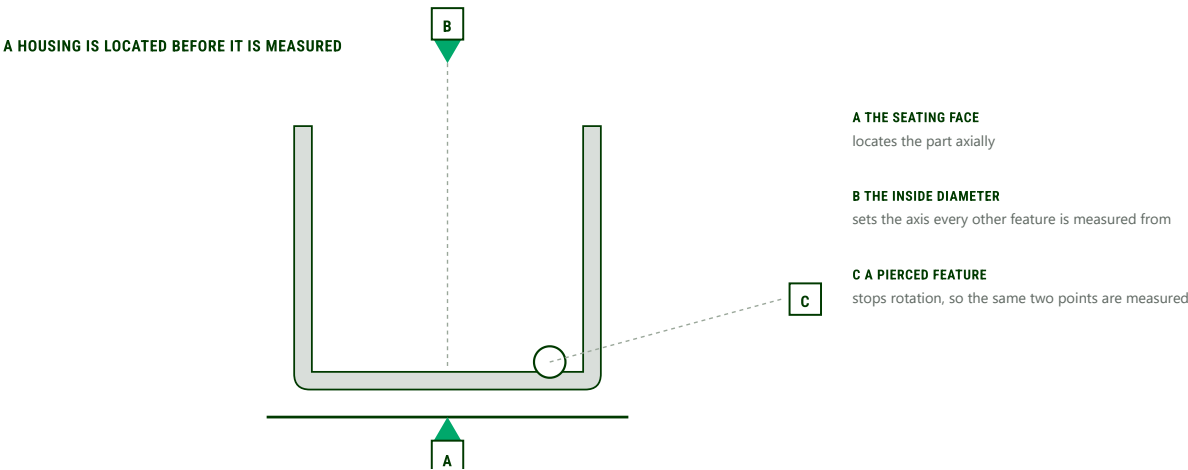


FIG 2.1 The datum scheme on a drawn housing. The seating face fixes the axial position, the inside diameter fixes the axis, and a pierced feature stops rotation.

Why the seating face, not the rim

The bottom of a drawn housing is a formed surface that seats against something in the assembly. The rim is a trimmed edge. Referencing to the rim puts the trim tolerance into every other measurement on the part.

Why the inside diameter

On a housing that receives a component, the inside diameter is functional and the outside is often clearance. Making the inside the axis datum means the tolerance stack is measured from the surface that matters.

Why a third datum is needed

With only two datums the part can still rotate. Any feature that is not rotationally symmetric - a hole, a flat, a notch - will then be measured in a different position every time.

One sentence that prevents the argument
Diameter to be measured at 20 mm above the inside bottom face, datum A, free standing, at any circumferential position.

CHOOSING THE PRIMARY DATUM - AND WHAT THE CHOICE COSTS

DATUM A	WHAT IT MEANS IN PRACTICE	WHEN IT IS THE RIGHT CHOICE
The seating face	The part sits as it will in the assembly. Trim and rim variation stay out of the stack.	Almost always. This is the default for a housing.
The trimmed rim	The trim tolerance enters every other measurement, including the height.	Only when the rim is the functional locating face.
The outside diameter	The part is located by its outside, so wall variation appears as run-out.	When the housing is pressed into a bore and nothing else locates it.

CHAPTER 03

THE SEVEN CALL-OUTS THAT DECIDE A HOUSING

Seven features carry almost all of the risk on a drawn housing. Each one has a typical as-drawn result and a specific operation that buys it tighter.

FEATURE - TYPICAL AS DRAWN - WHAT BUYS IT TIGHTER

FEATURE	TYPICAL AS DRAWN	BOUGHT TIGHTER BY	WHAT IT COSTS
Inside diameter	IT9 - IT11	Sizing or ironing pass	One more die and one more handling step.
Outside diameter	IT10 - IT12	Sizing ring	Same die as the sizing pass where both are sized together.
Wall thickness	about +/- 5% of nominal	Ironing	A separate die, a lubricant step and less remaining ductility.
Roundness	0.03 - 0.10 mm	Sizing pass	Included in the sizing operation, if it is planned at the outset.
Coaxiality, ID to OD	0.05 - 0.15 mm	Single-pass sizing from one datum	Needs a tool designed for it, not a tool that happens to do it.
Height	+/- 0.2 - 0.5 mm	Trim fixture referenced to datum A	A trim die and a fixture. Cheap compared with the alternative.
Bottom flatness	0.10 - 0.30 mm	Coining or machining	A separate operation, and often the most expensive line on the list.

Coaxiality is not one thing

ISO 1101 and ASME Y14.5 define concentricity differently, and a drawing that names neither will be interpreted both ways. State the standard. If what you actually need is "the wall is even all round", roundness plus wall thickness is the honest call-out.

The cheapest tolerance is the one you do not need

Before tightening a call-out, ask what the assembly does with that surface. On most housings only one of the seven is genuinely functional; the rest are drawing habit, and each one is paid for at every operation.

CHAPTER 04

WHAT THE PROCESS DOES TO THE WALL

A drawn wall is thinnest where it had to bend and stretch the most, which is just above the bottom radius. A wall tolerance written without knowing that will be argued about on the first sectioned sample.

WHERE THE WALL ACTUALLY GOES

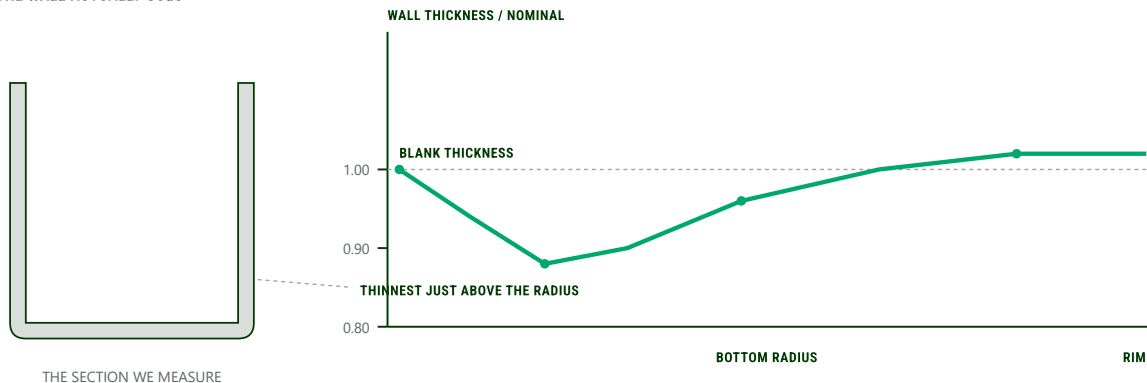


FIG 4.1 Typical wall thickness distribution on a straight draw, expressed as a fraction of blank thickness. The dip sits just above the bottom radius; the wall recovers toward the rim.

The dip is predictable

It is not a defect. It is where the blank was pulled over the die shoulder and the material had to stretch rather than flow. The deeper the draw and the tighter the radius, the deeper the dip.

Ironing flattens the profile

An ironing pass pulls the wall through a ring and equalises thickness along the length. That is what it is for. It does not remove the dip entirely, and it costs a die.

How to write a wall tolerance that works

State a nominal wall and a tolerance, then state where it applies. If the wall must be uniform, say so and accept ironing. If only one zone is functional - typically the seal area near the rim - say which zone and let the rest run.

Why a sectioned sample, not an ultrasonic reading

Ultrasonic thickness gives a fast number at one point. A sectioned sample gives the whole profile in one picture, and the profile is what the argument is about. Use both: ultrasonic for production control, section for first article.

CHAPTER 05

SPRINGBACK, AND THE SIZING PASS

When the punch withdraws, the wall relaxes elastically. The part that comes out of the die is slightly larger than the die that made it, and the difference is not the same around the circumference.

It is small, and it is not zero

Springback on a drawn diameter is a fraction of a percent of the diameter, which sounds negligible until the call-out is IT9. On a 40 mm housing, a tenth of a percent is 0.04 mm - most of an IT9 band.

It varies with the material

Austenitic stainless springs back more than deep drawing steel and much more than aluminium. Electrical pure iron, which is often used for solenoid housings, behaves differently again because it is a low ductility material.

It varies around the part

Because the wall thickness varies around the circumference, the elastic recovery does too. This is why a sizing pass improves roundness as well as diameter.



FIG 5.1 A drawn wall after release. The surface records the path the material took; the diameter records how much of it sprang back.

THREE WAYS TO DEAL WITH SPRINGBACK

APPROACH	HOW IT WORKS	WHEN TO CHOOSE IT
Size the part	A sizing die holds the diameter at the required value after the draw has released.	Whenever the diameter or roundness is functional. This is the normal answer.
Overdraw and let it come back	The die is cut slightly small and the part springs into tolerance.	Only for a stable, single material and a stable wall. It is sensitive to coil-to-coil variation.
Widen the tolerance	Accept the variation and design the assembly around it.	When the diameter is clearance and something else locates the part.

CHAPTER 06

INSPECTION METHODS AND THEIR LIMITS

Every method proves something slightly different. Choosing the method is part of choosing the tolerance, because a call-out that no available method can resolve is not a requirement - it is a negotiation.

METHOD - WHAT IT PROVES - WHERE IT FAILS

METHOD	WHAT IT PROVES	WHERE IT FAILS
Bore gauge	Diameter at a specific height, quickly, in production.	Gives one diameter. Says nothing about roundness or taper.
Air gauge	Diameter and some form error, very fast, on a clean surface.	Needs a dedicated plug per diameter. Sensitive to oil and burrs.
CMM, contact	Diameter, roundness, coaxiality and position, all against a stated datum.	Slow, and the probe force can deflect a thin wall if the part is not supported.
Optical or vision	Rim profile, hole position, earing, and features that contact methods struggle with.	Line of sight only. Cannot see inside a deep housing.
Ultrasonic thickness	Wall thickness at a point, non-destructively.	Reads through the coating and the coupling; needs calibration on the actual material.
Sectioned sample	The whole wall profile, thinning pattern and radius in one view.	Destructive. One part per batch, chosen deliberately.
Surface plate and indicator	Flatness, height and run-out against a physical datum.	Needs the right support. A thin part will follow the plate.

Matching the method to the call-out

Diameter: bore gauge in production, CMM on first article.
Roundness and coaxiality: CMM only. Wall: ultrasonic in production, section on first article. Rim and holes: optical.
Writing this pairing into the control plan is what makes a PPAP defensible.

Guard banding

Where a measurement method consumes a large share of the tolerance - a thin wall on a contact CMM, for example - production is held inside a narrower internal band so that the customer's reading stays inside the drawing. Ask for that band in the control plan.

A WORKED TOLERANCE BLOCK

This is the same 40 mm housing used in WP-01, written the way it should reach the shop floor. Nominal values are the example part; the structure is the point.

TOLERANCE BLOCK - 40 MM INSIDE DIAMETER DRAWN HOUSING

FEATURE	NOMINAL	TOLERANCE	DATUM	QUALIFIER
Inside diameter	40.00	+0.05 / 0	B	Measured 20 mm above datum A, free standing.
Outside diameter	42.00	+/- 0.10	B	Same height as the inside diameter.
Wall thickness	1.00	+/- 0.05	-	Sectioned sample on first article; ultrasonic in production.
Roundness	-	0.05	B	In one plane, at the inside diameter measuring height.
Coaxiality, ID to OD	-	0.10	A-B	ISO 1101. Datum axis established from the inside diameter.
Overall height	78.00	+/- 0.30	A	To the trimmed rim, measured at the highest point.
Bottom flatness	-	0.20	A	Over the seating area only, after the part has stabilised.

What the qualifier column is doing

It removes the four arguments in chapter 9 before they happen. Every qualifier answers one question: at what height, from what datum, under what support, measured how. None of them costs anything to write.

Where this block has to appear

On the drawing, in the title block notes, and in the control plan - in the same words. A tolerance that exists on the drawing but not in the control plan is not being managed; it is being discovered.

CHAPTER 08

FIRST ARTICLE AND PPAP

The tolerance block is a claim. The first article is the evidence. What sits between them is the dimensional report, and it is worth specifying what goes in it.

Sample size and selection

One part from one cavity proves almost nothing about a drawn housing, because thickness and roundness vary around the circumference. State how many parts, from how many coils, and whether they are consecutive or spread across the run.

Report every call-out, including the easy ones

A dimensional report that shows only the tight features hides the ones that are drifting. Every call-out in the tolerance block should appear, with the actual value and the deviation.

Attach the material certificate

Tolerances cannot be discussed without the coil. Grade, heat number and mechanical properties belong with the dimensional report, not in a separate email.

What to ask for at PPAP

- Dimensional report against every call-out, with the measuring method named.
- Sectioned sample, photographed, showing the wall profile.
- Material certificate with heat number.
- Control plan naming the method and frequency per feature.
- Process flow, so the operations that create each tolerance are visible.
- Any deviation, raised before submission rather than after.

The deviation conversation

Every drawn part programme has at least one call-out that the process cannot hold as written. The only question is whether that is discovered at first article or at the customer's incoming inspection. A supplier who raises it early is not weaker; a supplier who does not is a risk.

SAMPLE SIZE - WHAT IT CAN AND CANNOT SHOW

SAMPLE	WHAT IT SHOWS	WHAT IT HIDES
One part, one position	That the tool can produce one good part on one occasion.	Circumferential variation, coil-to-coil variation, and drift.
One part, four positions	Whether the wall and roundness vary around the circumference.	Whether the next part will be the same.
Five consecutive parts	Whether the process repeats within a run.	Whether it repeats across coils or shifts.
Parts from three coils	Whether the result survives a material change, which is the most common source of drift.	Long-term drift, which only production data shows.

THE FOUR ARGUMENTS THAT REPEAT

These come back on programme after programme, and each one is settled by a sentence that costs nothing to add at the drawing stage.

ARGUMENT - CAUSE - THE SENTENCE THAT ENDS IT

THE ARGUMENT	WHAT ACTUALLY HAPPENED	THE SENTENCE THAT PREVENTS IT
"Your diameter is out"	Two inspectors measured at two different heights on a tapered wall.	Diameter measured at 20 mm above datum A.
"The wall is thin"	One reading came from the zone just above the radius, the other from near the rim.	Wall tolerance applies to the cylindrical wall above the bottom radius.
"It is not round"	The part was clamped in a fixture while being measured.	Measured free standing, supported on datum A only.
"The height varies"	Earing, plus a trim referenced to the outside instead of the bottom.	Height measured to the trimmed rim, trim referenced to datum A.
"It was fine last month"	The coil changed. Drawability and springback moved with the material.	Material changes require notification and a new first article.
"The sample was in tolerance"	The sample was measured by hand, production is measured by gauge, and the two disagree.	One measurement method, named in the control plan, used by both sides.

Why this is worth reading twice

None of these four is a quality failure. In every case the part was within the tolerance that was actually written; the tolerance that was written was just not specific enough to be measured. The cost lands on the programme as rework, sorting or a concession.

The measurement agreement

For a new programme, agree the measurement method for each functional call-out before the first article, not after it. A one-page measurement plan signed by both sides removes the entire category of argument above.

CHAPTER 10

BEFORE THE DRAWING IS RELEASED

Twelve questions. If all twelve have an answer on the drawing, the first article becomes a formality rather than a negotiation.

On the drawing

- Are datums A, B and C defined, and is A the seating face?
- Is the measuring height stated for every diameter?
- Is the support condition stated for thin wall features?
- Does the wall tolerance say which zone it applies to?
- Is the geometric standard named - ISO 1101 or ASME Y14.5?
- Is the general tolerance class named - ISO 2768-1 class m or other?

Behind the drawing

- Is there a measurement method named for each functional call-out?
- Is the sample size and selection stated for the first article?
- Is the material grade and temper stated, with the certificate required?
- Is it stated whether the finished part must be magnetic?
- Is the PPAP level named, and the submission date?
- Is it agreed who pays if a call-out has to be relaxed after first article?

Send a drawing

We will return a DFM note naming the call-outs the process cannot hold as written, what each would cost to hold, and what the schedule looks like either way. That note is the basis of the quotation.

shawn@balford.net · balford.net

THE TWELVE ANSWERS, AS THEY APPEAR ON A RELEASED DRAWING

QUESTION	WHERE THE ANSWER LIVES
Datums A, B and C	Datum feature symbols in the drawing views, with A on the seating face.
Measuring height	Note in the title block, referenced by every diameter call-out.
Support condition	Note in the title block, or on the control plan for thin wall features.
Wall zone	Hatched zone on the section view, plus a note naming the boundary.
Geometric standard	Title block note: ISO 1101 or ASME Y14.5, with the edition.
General tolerance class	Title block note: ISO 2768-1 class m, or the class you have chosen.
Measurement method per feature	Control plan, agreed before first article.
Sample size and selection	Purchase order or quality agreement, not the drawing.

Balford.

DEEP DRAWN AND STAMPED METAL PARTS, FROM TOOLING TO PPAP

Zhejiang Balford Mechatronics makes deep drawn housings, cups and shells for sensor, solenoid valve and automotive programmes, with in-house tooling, drawing, ironing and inspection.

Process

Deep drawing, redrawing, ironing, progressive die stamping, secondary machining and surface treatment.

Materials

Deep drawing steel, stainless 304 and 316L, ferritic stainless, electrical pure iron, aluminium, copper and brass.

Quality

First article, PPAP, material certificates, and IMDS data where the programme requires it.

Send a drawing

shawn@balford.net
balford.net

This whitepaper

WP-02 Drawn Housing Tolerances, edition 2026. Companion to WP-01 Deep Draw Design Handbook. Illustrations are representative of the process and are not drawings of any customer part.