

# Cellborne CNC Design Guidelines

## Purpose & scope

Design parts to these guidelines and they can be machined at Cellborne quickly, at the quoted price, without a design-for-manufacturing (DFM) back-and-forth. The rules apply to metal and plastic parts machined on Cellborne's 5-axis equipment.

- **Recommended** values are what to design to by default; they keep cost and lead time low.
- **Minimum** values are achievable but add cost, risk or inspection time; call them out on the drawing.
- Anything outside these limits is not refused outright. Send it for review and Cellborne will confirm feasibility before quoting.
- Dimensions are in inches with metric equivalents. When a drawing and a 3D model disagree, the drawing governs.

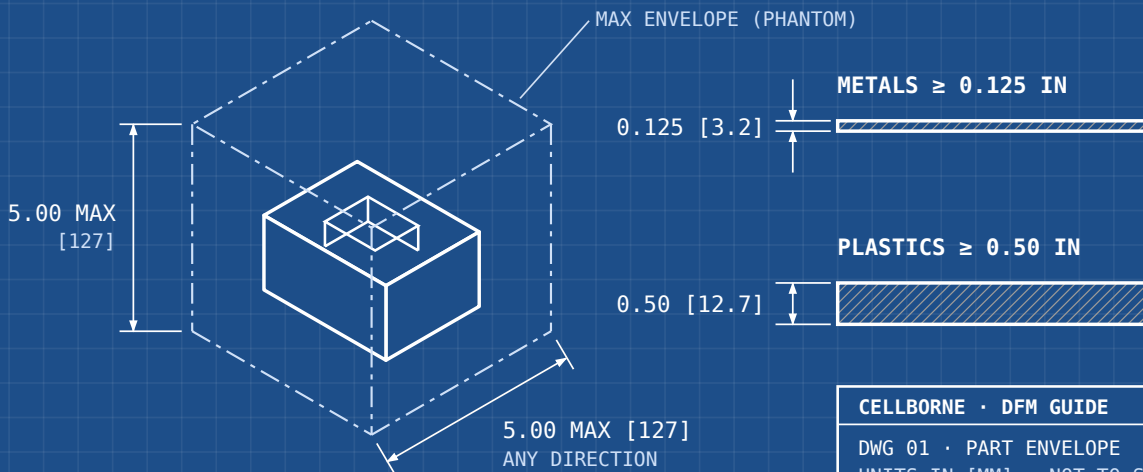
## Cellborne capabilities

Cellborne machines small precision parts, up to 5 in (127 mm) in the longest dimension, on an automatic 5-axis cell with 100% 5-axis CMM inspection.

| Parameter                                  | Metals                                                       | Plastics         |
|--------------------------------------------|--------------------------------------------------------------|------------------|
| Process                                    | 5-axis CNC milling (Brother Speedio M200Xd1-5ax, 16,000 rpm) | Same             |
| Maximum longest dimension                  | 5 in (127 mm)                                                | 5 in (127 mm)    |
| Minimum shortest dimension                 | 0.125 in (3.2 mm)                                            | 0.5 in (12.7 mm) |
| Minimum feature size (suitable geometries) | 0.04 in (1.0 mm)                                             | 0.04 in (1.0 mm) |
| Inspection                                 | 100%, automatic 5-axis coordinate measurement                | Same             |
| Design data                                | 3D model required; drawing optional; MBD and GD&T supported  | Same             |

## EVERY PART FITS A 5 IN ENVELOPE

Longest dimension 5.00 in max; shortest 0.125 in metals, 0.50 in plastics



The longest dimension of the finished part must fit within 5 in; its thinnest section must meet the material minimum.

- **Why plastics need 0.5 in minimum:** plastics deflect and move under clamping and cutting heat, so very thin or small plastic parts cannot be held and cut accurately.

## Materials

Choose aluminum 6061-T6 unless the application needs something else; it machines fastest and is the lowest-cost option. All materials are sourced traceable from reputable suppliers.

| Group                | Supported grades                          | Design notes                                                                                                                                          |
|----------------------|-------------------------------------------|-------------------------------------------------------------------------------------------------------------------------------------------------------|
| Aluminum             | 6061-T6/T6511, 7075-T6, MIC-6             | 6061 is the default. 7075 for high strength-to-weight. MIC-6 cast plate for flat, stable plates and fixtures.                                         |
| Carbon & alloy steel | 1018, 1215, A36, 4130, 4140, 4140PH, 4340 | 1215 is free-machining (lowest cost steel). 4130/4140/4340 for strength; specify heat-treat condition. Steels rust; plan a coating or oil.            |
| Stainless steel      | 303, 304/304L, 316/316L, 17-4PH, 410, 416 | 303 and 416 are free-machining. 316 for corrosion resistance. 17-4PH for strength + corrosion. Expect higher cost and longer lead time than aluminum. |

| Group                | Supported grades                                                                                                          | Design notes                                                                                                                                                              |
|----------------------|---------------------------------------------------------------------------------------------------------------------------|---------------------------------------------------------------------------------------------------------------------------------------------------------------------------|
| Engineering plastics | Acetal homopolymer (Delrin 150), Acetal, PTFE (Teflon), PEEK, Acrylic, PVC, ULTEM, UHMW-PE, Polycarbonate (PC), HDPE, ABS | Acetal is the default: stiff, stable, machines cleanly. PTFE, UHMW and HDPE are soft and move with temperature; loosen tolerances. PEEK and ULTEM for heat and chemicals. |

- Specify the exact grade and temper on the drawing (for example "Al 6061-T6"), not just "aluminum".
- Materials not listed can be reviewed on request.

## Tolerances & surface finish

Metal parts hold  $\pm 0.005$  in ( $\pm 0.13$  mm) on size and location and  $\pm 0.002$  in ( $\pm 0.05$  mm) on hole size; flat metal faces reach 63  $\mu$ in Ra (1.6  $\mu$ m). Tighten only the features that need it.

| Feature                                   | Metals                                        | Plastics                        |
|-------------------------------------------|-----------------------------------------------|---------------------------------|
| Size / location (standard)                | $\pm 0.005$ in ( $\pm 0.13$ mm)               | $\pm 0.010$ in ( $\pm 0.25$ mm) |
| Hole size                                 | $\pm 0.002$ in ( $\pm 0.05$ mm)               | $\pm 0.010$ in ( $\pm 0.25$ mm) |
| Flat-surface finish, as machined          | 63 $\mu$ in Ra (1.6 $\mu$ m)                  | Not applicable.                 |
| Contoured / side-wall finish, as machined | 63–125 $\mu$ in Ra (1.6–3.2 $\mu$ m), typical | Not applicable.                 |

Contoured and side-wall finishes depend on part geometry and tooling; the range shown is typical.

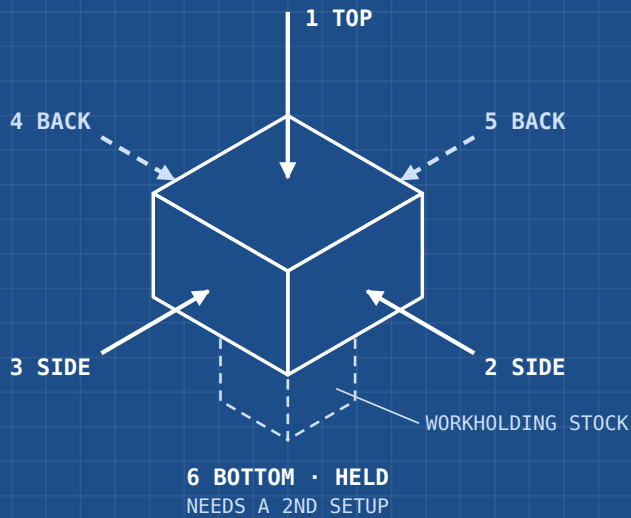
- **Untoleranced dimensions** are machined to the standard size/location tolerance above. State a title-block default on drawings so there is no ambiguity.
- **Tolerance only what matters.** Each tight tolerance adds machining passes and inspection time. Mating fits, bearing bores and datum faces deserve them; cosmetic outer faces do not.
- **GD&T:** define datums on faces that are machined in the same setup as the critical features, and on surfaces large enough for the CMM to probe ( $\geq 0.125$  in / 3 mm wide).
- **Plastics** expand with temperature and relax after machining; PTFE, UHMW and HDPE may need looser limits than the table shows.
- **Finishes:** parts ship as machined. Note any cosmetic face on the drawing so tool marks and fixturing are kept off it.

## Feature & face accessibility

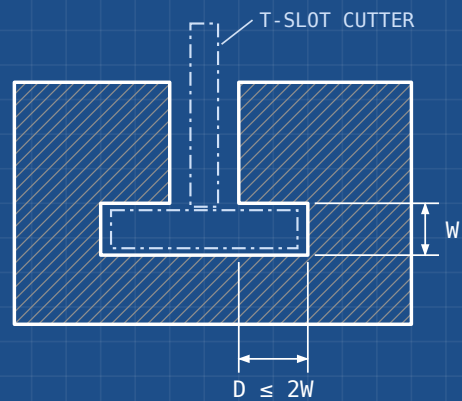
Every machined feature needs a straight line of sight for a rotating tool from one of the five faces the 5-axis machine can reach in a single setup. The sixth face is held by the fixture.

## FIVE FACES IN ONE SETUP; THE SIXTH IS HELD

Every feature needs a straight tool path from the top or one of the four sides



### UNDERCUT • SECTION A-A



CELLBORNE • DFM GUIDE

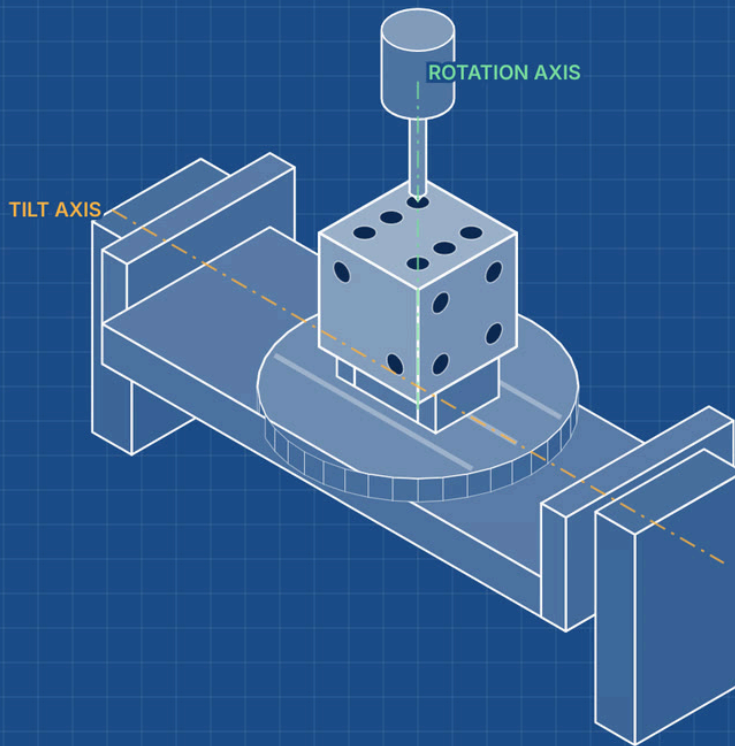
DWG 02 • FACE ACCESS  
NOT TO SCALE

Faces 1–5 are machined in one setup; the bottom face needs a second. Undercuts are cut with a standard T-slot or dovetail cutter.

- **Five faces, one setup.** Top and all four sides, plus angled faces and holes, can be machined without re-clamping. This keeps features on those faces accurately located to each other.
- **The held face.** The bottom of the stock is clamped. Parts that need features on all six faces require a second setup, which adds cost and loosens face-to-face location. Put critical relationships on the five accessible faces.

## 5-AXIS: TILT AND ROTATE THE PART, MACHINED FROM ABOVE

A part is machined on five faces in one setup; the clamped sixth face needs a flip.



### SETUP 2 OF 2

|          |      |
|----------|------|
| TILT     | 0.0° |
| ROTATION | 0.0° |

Features machined: 21 of 21

### 6 FACES, 2 SETUPS

Faces 1-5 in one clamping.  
Face 6 needed a flip.

CELLBORNE · DFM GUIDE  
ANIM 05 · 5-AXIS SETUPS

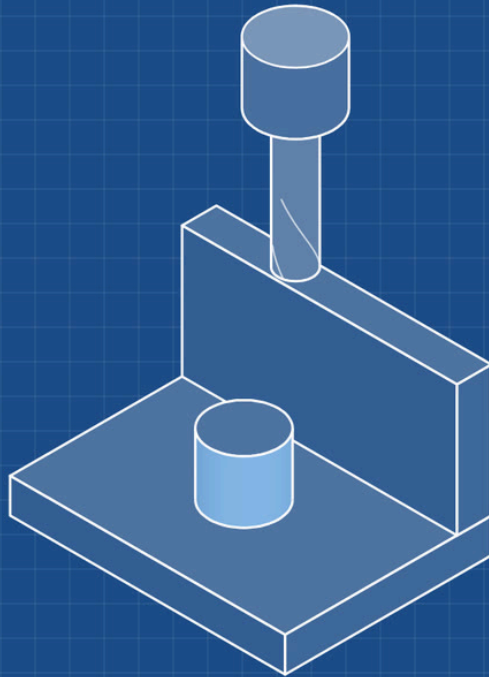
Animation: A part on a 5-axis trunnion tilts and rotates so a tool coming from above reaches five faces, then the part is flipped in the vise to machine the sixth. Watch it at [cellborne.com/design-guidelines](http://cellborne.com/design-guidelines).

On the 5-axis machine the part tilts and rotates under a cutting tool that only moves straight down, so five faces are machined in one clamping. The clamped sixth face needs the part flipped into a second setup.

- **Tool reach.** Keep the line from the spindle to the feature clear of the part itself. Deep features next to tall walls force longer, slower tools.

## A BOSS WITH A CLEAR PATH FROM ABOVE IS ONE SIMPLE CUT

The end mill must run parallel to the boss to cut its round wall.



### ACCESSIBLE BOSS

- 1 Tool comes straight down beside the boss
- 2 It circles the boss to cut the round wall
- 3 Retract. Nothing was in the way.

### CLEAR PATH FROM ABOVE

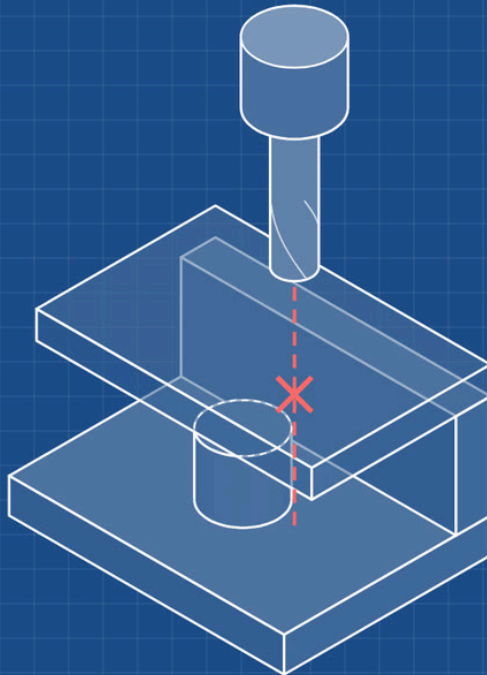
Cut in one setup with a standard end mill

CELLBORNE · DFM GUIDE  
ANIM 03 · ACCESSIBLE BOSS

Animation: An end mill comes straight down beside an open boss and circles it to cut the round wall. Watch it at [cellborne.com/design-guidelines](http://cellborne.com/design-guidelines).

## A SHELF OVER THE BOSS BLOCKS THE ONLY TOOL PATH

The end mill must run parallel to the boss to cut its round wall.



### OBSCURED BOSS

- 1 Tool comes straight down beside the boss
- 2 It hits the shelf before reaching the boss
- 3 No other straight path cuts the round wall

### CANNOT BE MACHINED AS DRAWN

Fix: move the boss out from under the shelf, or make the shelf a separate part.

CELLBORNE · DFM GUIDE  
ANIM 04 · OBSCURED BOSS

Animation: The same boss under an overhanging shelf; the end mill hits the shelf and cannot reach the boss. Watch it at [cellborne.com/design-guidelines](https://cellborne.com/design-guidelines).

To cut a boss's round wall, the end mill has to run parallel to the boss. Keep that straight path clear: a shelf, lip or overhang above the boss blocks it, and the boss can't be machined as drawn.

- **Undercuts.** Avoid them. Where needed, use standard T-slot or dovetail profiles, keep undercut depth  $\leq 2\times$  the slot width, and leave open space opposite the undercut of at least  $4\times$  its depth so the cutter head can enter.
- **Enclosed or internal cavities** that no straight tool can reach cannot be machined; split the part into machinable pieces and assemble them, or consider 3D printing.
- **Angled features.** Holes and faces at compound angles are fine on 5-axis. Give them a flat entry face perpendicular to the hole axis so drills don't walk.
- **Workholding stock.** Allow material below the part for clamping, which is removed afterward.

## Minimum feature sizes

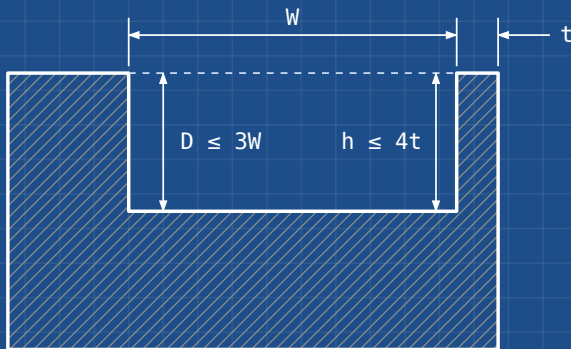
No feature can be smaller than 0.04 in (1.0 mm), Cellborne's minimum feature size; design to the recommended column whenever the part allows.

| Feature                           | Recommended                                                                                               | Minimum / limit             | Why it matters                                         |
|-----------------------------------|-----------------------------------------------------------------------------------------------------------|-----------------------------|--------------------------------------------------------|
| Wall thickness, metals            | $\geq 0.06$ in (1.5 mm)                                                                                   | 0.04 in (1.0 mm)            | Thin walls chatter, deflect and lose tolerance.        |
| Wall thickness, plastics          | $\geq 0.10$ in (2.5 mm)                                                                                   | 0.06 in (1.5 mm)            | Plastics flex and warp from cutting heat.              |
| Unsupported wall height           | $\leq 4 \times$ wall thickness                                                                            | $8 \times$ wall thickness   | Tall thin walls vibrate; add ribs or thicken the base. |
| Internal (vertical) corner radius | $\geq 1/3$ of pocket depth, and slightly larger than a standard tool radius (e.g. 0.130 in, not 0.125 in) | 0.02 in (0.5 mm)            | Round tools cannot cut square inside corners.          |
| Pocket floor-to-wall fillet       | 0 (sharp) or a standard ball/bull radius                                                                  | —                           | Sharp floor corners are cheapest.                      |
| Pocket / cavity depth             | $\leq 3 \times$ pocket width                                                                              | $6 \times$ pocket width     | Deep pockets need long, slow tools.                    |
| Hole diameter                     | <a href="#">Standard drill size</a> , $\geq 0.125$ in (3 mm)                                              | 0.04 in (1.0 mm)            | Standard drills are fast and accurate.                 |
| Hole depth                        | $\leq 4 \times$ diameter                                                                                  | $10 \times$ diameter        | Deep holes risk drill breakage and wander.             |
| Thread size (tapped)              | $\geq \#4\text{-}40$ / M3                                                                                 | #2-56 / M2                  | Small taps break easily.                               |
| Thread depth                      | $1.5 \times$ nominal diameter                                                                             | $3 \times$ nominal diameter | Engagement beyond $1.5 \times$ adds little strength.   |
| Bosses, pins, ribs                | Diameter or width $\geq 0.08$ in (2 mm), height $\leq 4 \times$ width                                     | 0.04 in (1.0 mm)            | Slender features bend under cutting load.              |

## ROUND TOOLS LEAVE ROUND INSIDE CORNERS

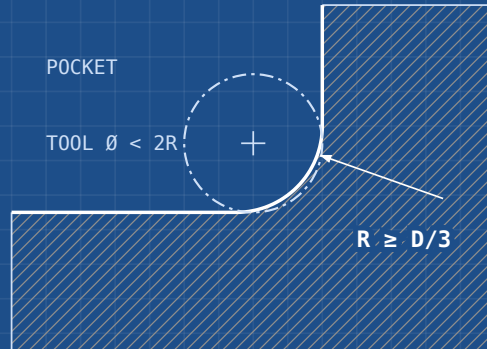
Pocket depth  $\leq 3 \times$  width · wall height  $\leq 4 \times$  thickness · corner radius  $\geq 1/3$  depth

SECTION A-A



SHARP FLOOR-TO-WALL CORNERS ARE FINE

DETAIL B · TOP VIEW



CELLBORNE · DFM GUIDE

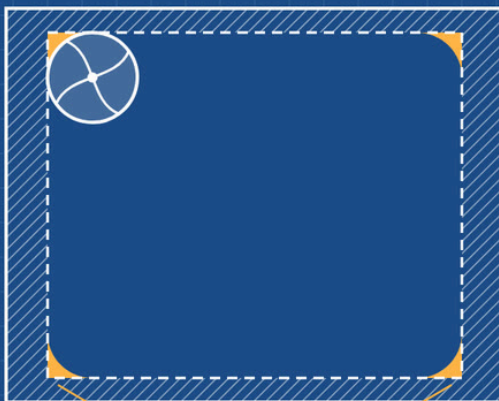
DWG 03 · POCKETS & WALLS  
NOT TO SCALE

Deep narrow pockets and tall thin walls force long, slow tools; a generous inside radius lets a larger tool cut the corner.

## A ROUND TOOL CANNOT CUT A SQUARE INSIDE CORNER

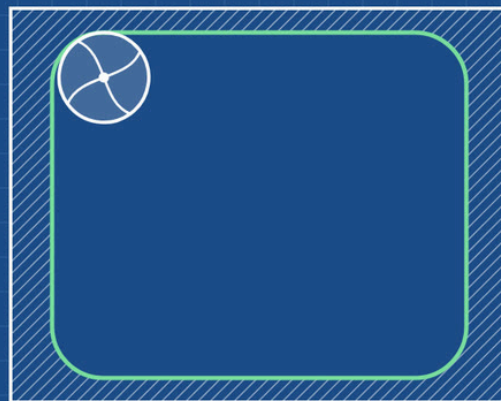
Top view of a pocket. Dashed line = the shape as designed; circle = the spinning end mill.

AS DRAWN: SHARP CORNERS



UNCUT MATERIAL IN EVERY CORNER

DESIGN FIX: CORNER  $R > \text{TOOL } R$

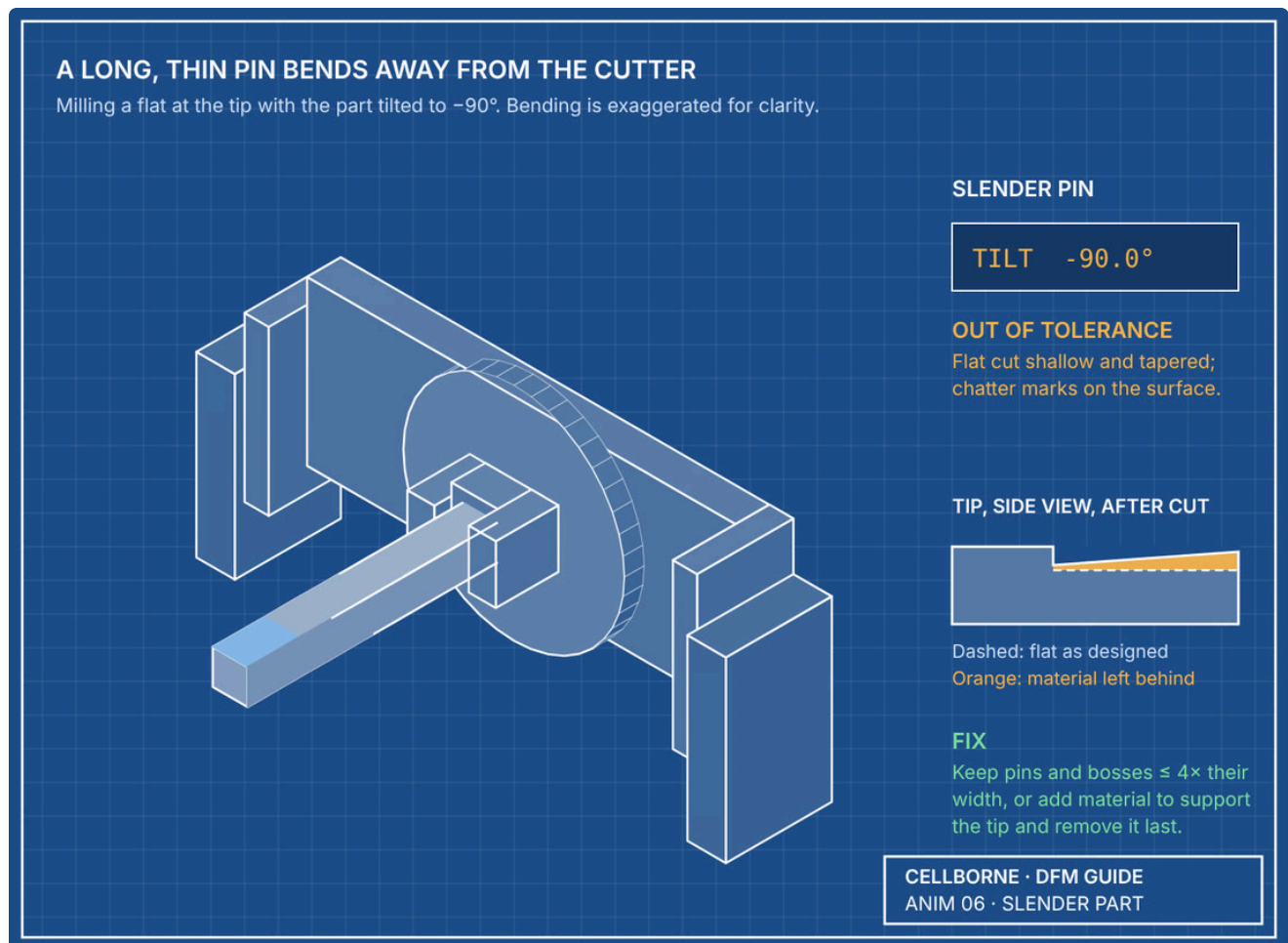


TOOL FITS, CORNER CUT CLEAN

CELLBORNE · DFM GUIDE  
ANIM 01 · INSIDE CORNERS

Animation: A round end mill leaves uncut material in sharp inside corners; a radiused corner cuts clean. Watch it at [cellborne.com/design-guidelines](http://cellborne.com/design-guidelines).

An end mill is round, so it always leaves a radius in an inside corner. Design that radius in, slightly larger than the tool's, and the corner comes out clean.



Animation: A slender pin tilted 90 degrees bends away from the end mill while a flat is cut at its tip, leaving the flat shallow and tapered. Watch it at [cellborne.com/design-guidelines](https://cellborne.com/design-guidelines).

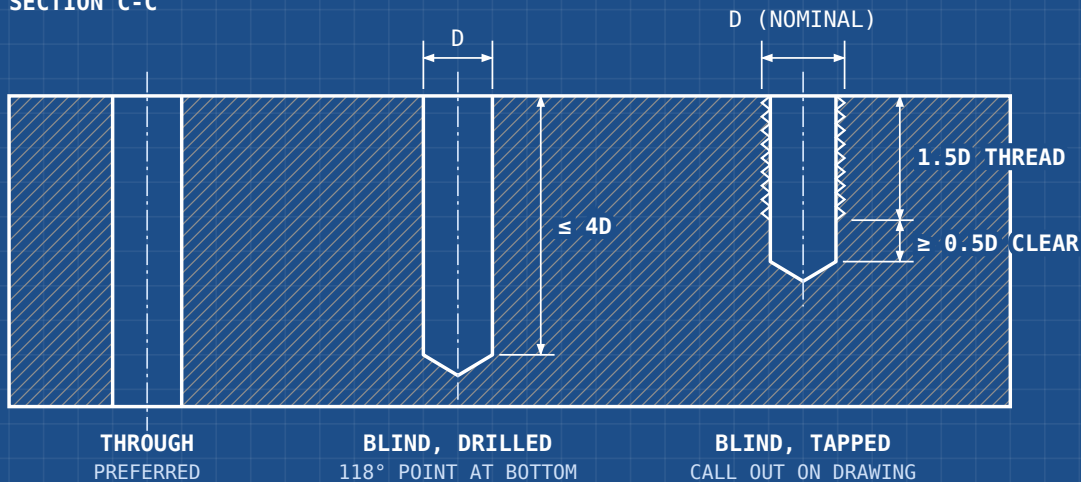
A long, thin feature acts like a diving board: the cutter pushes the tip away, it chatters, and it springs back with material left on. Keep pins and bosses within  $4\times$  their width, or design support material near the tip.

## Holes and threads

## BLIND HOLES END IN A 118° POINT; TAPS NEED ROOM BELOW

Hole depth  $\leq 4 \times D$  · thread depth  $1.5 \times D$  · at least  $0.5 \times D$  unthreaded below a blind tap

### SECTION C-C



CELLBORNE · DFM GUIDE

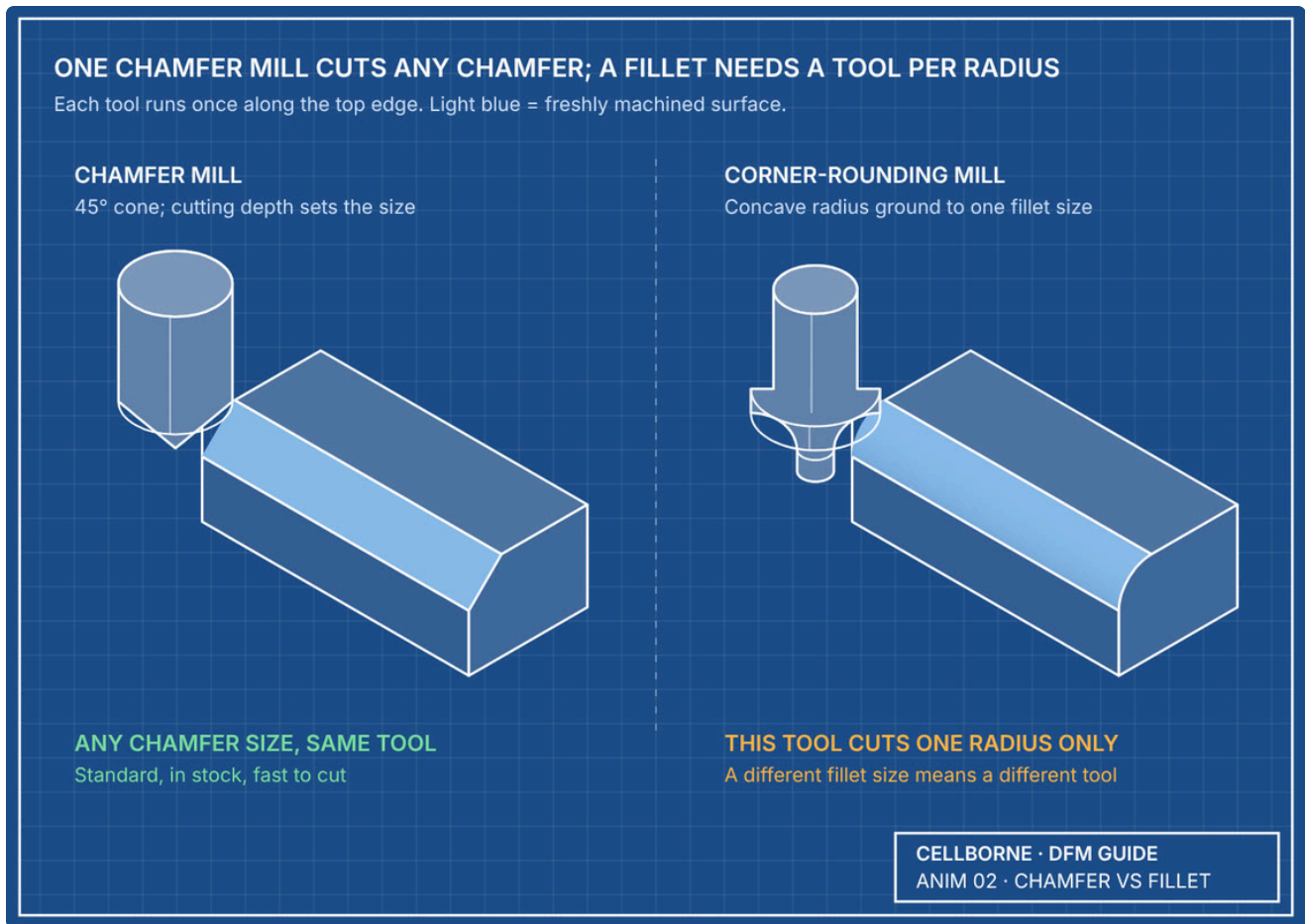
DWG 04 · HOLES & THREADS  
NOT TO SCALE

Through-holes are cheapest and most accurate; a blind tapped hole needs drilled depth past the last full thread.

- Blind holes have a 118° drill-point cone at the bottom; model them that way, or call out a flat bottom (milled, slower).
- Leave unthreaded depth below a blind tapped hole of at least  $0.5 \times$  the nominal diameter so the tap does not bottom out.
- Specify threads by callout (for example "1/4-20 UNC-2B  $\times$  0.38 deep"), not by modeled helix. Modeled threads are ignored.
- Through-holes are cheaper and more accurate than blind holes; use them where function allows.
- Avoid holes that break out on a curved or angled face; add a spot face or flat.

## Edges and corners

- External edges are deburred; call out a chamfer (e.g. 0.010–0.020 in  $\times$  45°) rather than an external fillet, which takes a special tool.
- Keep all internal radii in a part the same size where possible so one tool cuts them all.



Animation: A chamfer mill and a corner-rounding mill each run along a top edge; the chamfer mill cuts any chamfer size, the corner-rounding mill only its own radius. Watch it at [cellborne.com/design-guidelines](https://cellborne.com/design-guidelines).

One chamfer mill cuts any chamfer size by changing its depth. A corner-rounding mill cuts only the radius it was ground to, so each fillet size needs its own tool.

## Text, cost drivers & pre-RFQ checklist

### Text and markings

- Laser marking is faster and cheaper than engraving; choose it where you can.
- Engrave (recessed) rather than emboss (raised); engraving needs only the text cut, embossing needs everything around it removed.
- Use a bold sans-serif font, stroke width  $\geq 0.04$  in (1.0 mm), character height  $\geq 0.12$  in (3 mm), depth 0.010–0.020 in (0.25–0.5 mm).
- Place text on a flat face reachable from one of the five accessible faces.

### What drives cost (highest impact first)

1. Number of setups: features on the held sixth face, or that no single orientation can reach.
2. Tall, slender parts: they flex like a diving board under cutting load and may need extra setups or support material.

3. Tight tolerances and tight finishes beyond the standard values.
4. Small tools: tiny corner radii, deep narrow pockets, small threads.
5. Material: stainless, alloy steel and PEEK/ULTEM cost more to buy and to cut than 6061 or acetal.
6. Thin walls and slender features that force light, slow cuts.
7. External fillets: each radius needs its own special tool, so choose chamfers where you can.

#### **Before you submit an RFQ**

- ☐ Part fits within 5 in (127 mm) longest dimension, and meets the minimum shortest dimension (0.125 in metals / 0.5 in plastics).
- ☐ Material grade and temper are specified.
- ☐ No feature, wall or hole is smaller than 0.04 in (1.0 mm).
- ☐ Internal corner radii are  $\geq 1/3$  of pocket depth and not smaller than 0.02 in.
- ☐ Every feature is reachable in a straight line from the top or one of the four sides.
- ☐ Tight tolerances are applied only to functional features; a default tolerance is in the title block.
- ☐ Threads are called out on the drawing, not modeled.
- ☐ Cosmetic faces are marked.
- ☐ 3D model (STEP preferred) is attached; a drawing is included when tolerances, threads or finishes are critical.