

DFM Guide and Pre-submission Checklist

Design rules for CNC machining, 5-axis, sheet metal, 3D printing, injection molding and finishing, and what to send with your files. Every design still gets an engineer review.

Pre-submission Checklist

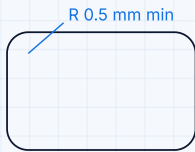
Send these with your files and the quote comes back faster, with fewer questions. An engineer reviews every order either way.

Send	What to include	Why it matters
3D model	STEP or IGES for machined parts. STEP, STL or 3MF for printed parts, with STEP preferred for metal. DXF or DWG also works for sheet metal	The model sets the geometry that is quoted and made
2D drawing	Tolerances, GD&T callouts, threads and inserts, and critical dimensions	Fine threads, custom pitches and inserts are only made when the drawing calls them out
Quantity	The prototype quantity, and expected production volumes if known	Setup and tooling costs are spread across the batch, so unit price depends on quantity
Material	The alloy or grade, for example 6061-T6 or 316L	Alloys finish differently. 2024 varies, and cast alloys typically cannot be hard anodized to a cosmetic standard
Finish	The finish spec, and the post-treatment dimensions for tight features	Anodize and plating add thickness, so dimensions are worked back from your post-treatment sizes
Masking	Threads, precision bores and sealing faces that must stay as machined	Without a note, coating can land on features that need to stay bare
Sheet metal details	Thickness, hardware, weld locations, and an assembly drawing for assemblies	The weld sequence affects final dimensions
3D print details	Main load direction and which faces are cosmetic	Prints are weaker in the layer direction, so orientation follows the load

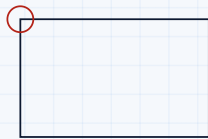
NDAs are available on request, before you share files.

CNC Machining

These four rules keep cost and lead time down. Parts that break them can usually still be made, and the design review will say how.



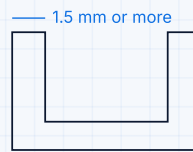
Radiused corner



Sharp corner

Internal Corners

- ✓ Use a radius of at least 0.5 mm. A third of the pocket depth is a safe start.
- ✗ Sharp inside corners. They need EDM or grinding and add lead time.



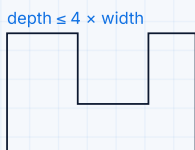
Stable wall



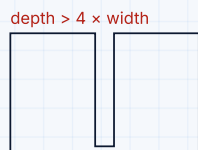
Thin wall

Wall Thickness

- ✓ Keep walls at 1.5 mm or more. They stay stable during cutting in most materials. Aluminium walls can go down to 0.8 mm.
- ✗ Thinner walls without flagging them. They flex under the cutter, and thin titanium walls need special fixturing.



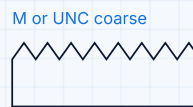
Open pocket



Deep narrow pocket

Pocket Depth

- ✓ Keep pockets no deeper than four times their width.
- ✗ Deep narrow pockets cut from one side. Long tools deflect, so tight tolerances get harder to hold. Cut them from several sides or in stages.



Standard thread



Needs a drawing note

Threads

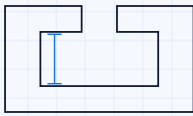
- ✓ Use standard coarse metric (M) or UNC threads. They are the fastest and cheapest.
- ✗ Leaving fine threads, custom pitches or Heli-Coil inserts off the drawing. All of them are available when the drawing calls them out.

Specifications, materials and finishes are on the [CNC machining page](#)

5-Axis Machining

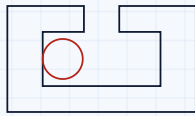
Treat these as guidance. If your geometry does not fit them, send it anyway and an engineer will advise.

undercut depth noted



Annotated undercut

no depth on the drawing

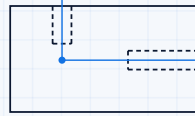


Unmarked undercut

Undercuts and Angled Features

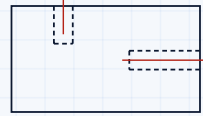
- ✓ Annotate undercut depths, angled bore directions and compound angles on the drawing, so the toolpath is planned before the quote.
- ✗ Leaving them unmarked, so geometry constraints turn up mid-job instead of at quoting.

linked across faces



Cross-face callout

each face on its own

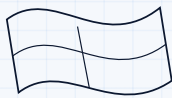


No cross-face callout

Tolerances Across Faces

- ✓ State tight positional relationships between faces explicitly on the drawing. Callouts of ± 0.02 mm or tighter across faces rely on the single setup.
- ✗ Leaving cross-face relationships off the drawing, with each face dimensioned on its own.

STEP model with Ra callouts



Whole surface defined

2D section views only

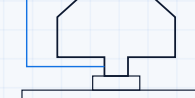


Gaps between sections

Freeform Surfaces

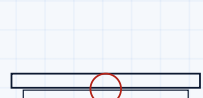
- ✓ Send the STEP model, because for compound-curved and freeform geometry it is the drawing. Add Ra callouts and any curvature tolerances for those faces.
- ✗ Relying on 2D section views. They do not capture enough of the surface.

clamping face



Secure clamping face

no face to clamp



Needs a custom fixture

Fixturing Access

- ✓ Give the part a secure clamping face that leaves every machined surface reachable on the rotary table.
- ✗ Very large flat parts, or parts with no good clamping face, sent without a flag. They may need a custom fixture.

Specifications, materials and finishes are on the [5-axis machining page](#)

Sheet Metal Fabrication

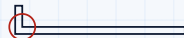
Treat these as guidance. Every design gets an engineer review whether or not it meets them.

$R \geq 1 \times t$, flange $\geq 4 \times t$



Radiused bend, long flange

$R < 1 \times t$, flange $< 4 \times t$

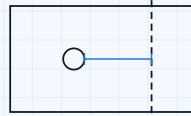


Sharp bend, short flange

Bend Radius and Flange Length

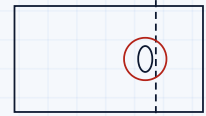
- ✓ Use an inside bend radius of at least $1 \times$ material thickness, and make flanges at least $4 \times$ thickness long so the press brake can grip them.
- ✗ Radii under $1 \times$ thickness, which risk cracking, and short flanges. Some alloys take tighter radii after DFM review.

$\geq 3 \times t + R$ from the bend



Hole clear of the bend

$< 3 \times t + R$ from the bend

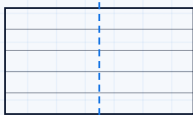


Hole deforms in forming

Holes Near Edges and Bends

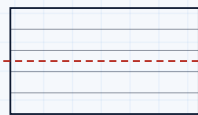
- ✓ Keep holes at least $2 \times$ thickness from an edge, and $3 \times$ thickness plus the bend radius from a bend.
- ✗ Holes closer than that, because they deform during forming. Mark critical dimensions on the drawing so problems are flagged before cutting.

bend across the grain



Lower cracking risk

bend along the grain



Higher cracking risk

Grain Direction

- ✓ Bend across the rolling grain, especially for tight-radius bends in aluminium and high-strength steel.
- ✗ Tight bends along the grain, where cracking is more likely. For bends in several directions, blank orientation is confirmed during DFM.

$\geq 3 \times OD$ from the bend



Fastener clear of the bend

$< 3 \times OD$ from the bend



Fastener against the bend

Hardware and Weld Placement

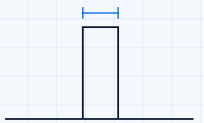
- ✓ Keep PEM fasteners and weld nuts clear of bends and edges, typically by $3 \times$ their outer diameter. Mark weld locations and access needs on the drawing.
- ✗ Fasteners crowded against a bend, or weld locations left off the drawing. The weld sequence affects final dimensions.

Specifications, materials and finishes are on the [Sheet metal fabrication page](#)

3D Printing

Guidance rather than fixed limits. If you are not sure a lattice, overhang or other feature will print, send the file and an engineer will advise.

at the process minimum



0.4 to 1.2 mm, by process

below the minimum

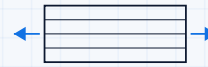


Warp or fails to build

Wall Thickness

- ✓ Keep walls at or above the process minimum: 0.4 mm for DMLS, 0.5 mm for SLA, 0.7 mm for SLS and MJF, and 1.2 mm for FDM.
- ✗ Thinner walls without flagging them in your brief. They are sometimes possible but risk failing or warping.

pulled along the layers



Strongest in X and Y

pulled across the layers

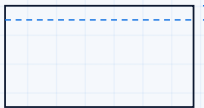


Weakest in Z

Build Orientation

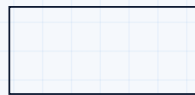
- ✓ Tell us the main load direction and which faces are cosmetic, and the part will be oriented to suit.
- ✗ Assuming a print is equally strong in every direction. All additive processes are anisotropic, and parts are weaker in Z, the layer direction, than in X and Y.

0.3 to 0.5 mm stock



Stock on critical face

no stock to machine



Printed to final size

Tight Tolerances

- ✓ Add 0.3 to 0.5 mm of stock to interface bores, sealing faces and threads that need tighter than ± 0.1 mm, so they can be CNC machined after printing.
- ✗ Modelling those surfaces at final size, which leaves no stock to machine. Print the geometry and machine the interfaces.

$\leq 45^\circ$ from vertical



No supports needed

$> 45^\circ$ needs supports



Needs removable supports

Overhangs on Metal Prints

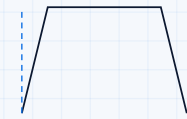
- ✓ DMLS and SLM need supports on overhangs more than 45° from vertical. Design those supports to be removable, with access ports to reach them.
- ✗ Supports that cannot be reached after printing. Expect a rougher finish where supports attach on interior surfaces.

Specifications, materials and finishes are on the [3D printing page](#)

Injection Molding

Guidance rather than fixed rules. Send the design either way: the DFM review checks every part, including multiple undercuts, tight tolerances and unusual materials.

1 to 3° draft



Drafted walls

0° draft

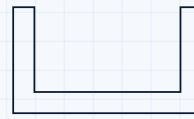


No draft

Draft Angles

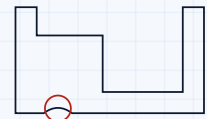
- ✓ Add 1 to 3° of draft to faces parallel to the pull direction so the part releases cleanly. Textured surfaces typically need 3 to 5°.
- ✗ Zero-draft faces. They are possible with side-pulls, but add tooling cost.

1.0 to 4.0 mm, uniform



Uniform wall

thick section sinks



Thick section

Uniform Wall Thickness

- ✓ Keep walls uniform, between 1.0 and 4.0 mm. Where a thick section is unavoidable, core it out to cut mass without losing stiffness.
- ✗ Thick sections. They cool unevenly and cause sink marks or warpage.

gate on a hidden face



Seam and gate agreed early

gate on the show face



Visible gate mark

Gates and Parting Line

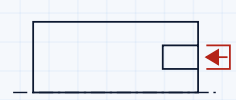
- ✓ Place gates on non-cosmetic surfaces where possible, and confirm the parting line location early.
- ✗ Gates on cosmetic faces, or a parting line left until late. Gate marks are visible, and the parting line sets where the seam sits and where flash can form.

forms through the parting line



Ejects straight off the tool

blocks straight ejection



Needs a side-pull or lifter

Undercuts and Side Features

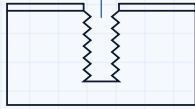
- ✓ Where possible, redesign undercuts to form through or to be reachable from the parting line.
- ✗ Undercuts that block straight ejection. They need side-pulls, lifters or collapsible cores, which add tooling cost.

Specifications, materials and finishes are on the [Injection molding page](#)

Surface Finishing

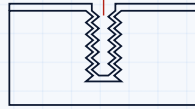
Check these four points before you finalise the drawing. The DFM review covers them too.

thread masked



Masked thread

coating in threads

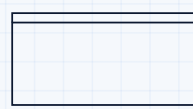


Coated thread

Masking

- ✓ Mark threads, precision bores and sealing faces that must stay as machined, for example "mask M5 holes".
- ✗ Leaving masking off the drawing. The DFM review flags features that likely need it, but a note up front saves time.

Ra 0.8 μm or better



Smooth substrate

marks show through

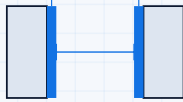


Rough substrate

Surface Finish

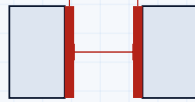
- ✓ Machine to Ra 0.8 μm or better before decorative anodize or bright plating, and put the finish on the drawing.
- ✗ Relying on the coating to hide machining marks. It follows the surface underneath.

bore machined oversize



Coated bore lands on the drawing

bore machined to final size

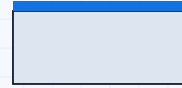


Coated bore ends undersize

Coating Buildup

- ✓ Machine tight-tolerance features to pre-treatment size and give post-treatment dimensions on the drawing. We work back from those.
- ✗ Ignoring buildup. Type II anodize adds about 2.5 to 12.5 μm per surface, Type III up to 38 μm and electroless nickel up to 50 μm .

even anodic layer



6061 and 7075 anodize well

uneven, patchy layer



2024 and cast alloys vary

Alloy Choice

- ✓ Call out the alloy on the drawing. 6061 and 7075 anodize well, and we will flag any process limits.
- ✗ Expecting every alloy to finish the same. 2024 varies, and cast alloys typically cannot be hard anodized to a cosmetic standard.

Specifications, materials and finishes are on the [Surface finishing page](#)

Ready to quote?

Upload your CAD and drawing at app.factorrem.co/getquotenow. To talk to an engineer, go to factorrem.co/contact.