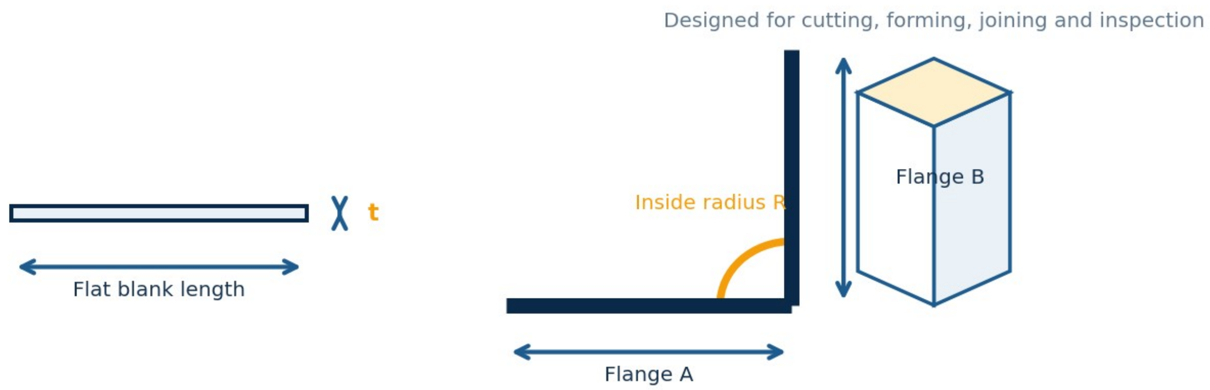
KINGSFORM METALWORKS

ENGINEERING TOOLKIT

SHEET METAL DESIGN GUIDE

Design rules for cutting, bending, hardware insertion, joining, finishing and inspection



DOCUMENT	VERSION	RELEASE DATE	LANGUAGE
KFR-008	1.0	2026-07-17	English

FORMAT	OWNER	CONTACT
A4 PDF	KingsForm Engineering	info@kingsformmetalworks.com

Engineering reference only. The released drawing, customer requirements and validated process remain controlling.

Scope and how to use this guide

A practical DFM reference for fabricated sheet metal parts, boxes, brackets, panels and welded assemblies.

Use this guide before the flat pattern, tolerances, cosmetic surfaces and hardware sequence are frozen.

This document connects product design intent with the realities of blanking, forming, hardware insertion, joining, finishing and inspection. It is written for designers, buyers and manufacturing engineers who need drawings that are clear enough to quote and robust enough to manufacture repeatedly.

Use it to	Do not use it to
Screen bend geometry, hole locations, reliefs and tool access.	Replace material data, shop bend tables or process qualification.
Define what belongs on the drawing and what should stay in the process plan.	Assume one universal K-factor, minimum radius or tolerance for every material.
Identify cost and quality risks before RFQ release.	Copy proprietary standard tables without a controlled source.

Recommended workflow

1. Define function, material, thickness, finish, annual volume and critical interfaces.
2. Choose the likely blanking and forming route before specifying tight tolerances.
3. Use shop-proven bend data for developed length, springback and minimum flange checks.
4. Confirm holes, hardware, welds and surface requirements are compatible with the bend sequence.
5. Inspect the final formed state using functional datums and a risk-based inspection plan.

Contents

02 SHEET METAL TERMINOLOGY AND PROCESS CHAIN

Sheet metal terminology and process chain

Use consistent geometry definitions before discussing tolerances or flat patterns.

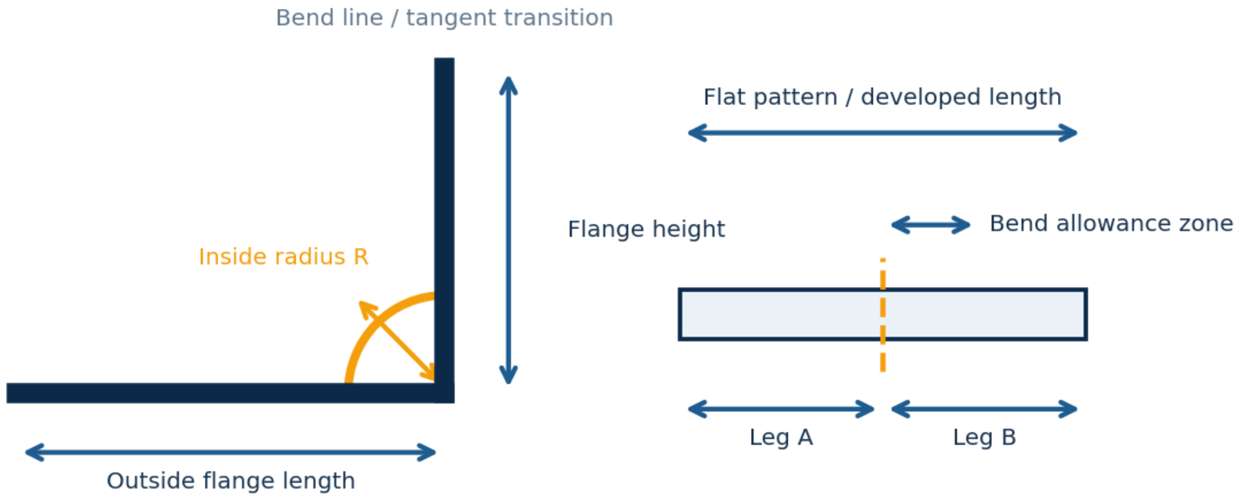


Figure 1. Formed-part and flat-pattern terminology used in this guide.

Term	Definition	Drawing implication
Sheet thickness T	Nominal material thickness before forming.	Specify grade, temper / condition and thickness tolerance source.
Inside radius R	Radius on the inside surface of the bend.	Do not mix inside radius with punch radius or outside radius.
Bend angle A	Angle through which the sheet is bent in this guide.	State the convention if using bend formulas or bend tables.
Bend line	Reference line in the flat pattern that represents the bend location.	Usually process information; final formed geometry should control.
Tangent point	Transition between straight flange and bend arc.	Use for hole-to-bend and relief-distance checks.
Flat pattern	Developed blank before forming.	Treat supplier-generated flats as process data unless contractually controlled.

Freeze the process route before tightening tolerances or cosmetic requirements

The process route is coupled: a decision made for finish, hardware or welding can change blank size, bend sequence, tooling access and inspection.

03 MATERIAL AND THICKNESS SELECTION

Choose material and thickness as a manufacturing system

Grade, temper, rolling direction, surface condition and thickness all affect forming and finish.

Material family	Design strengths	Forming concerns	Typical drawing details
Low-carbon steel	Good formability, weldability and cost efficiency.	Coating requirement, rust, springback variation and burr control.	Standard / grade, thickness, finish and coating system.
High-strength steel	Strength and gauge reduction potential.	Higher springback, force and cracking risk; larger radii may be required.	Exact grade, yield range and approved substitutions.
Stainless steel	Corrosion resistance and strong cosmetic options.	Higher work hardening, springback and tooling-mark sensitivity.	Grade, finish direction, protective film and weld-cleaning requirement.
Aluminum 5xxx	Good corrosion resistance and bendability in suitable tempers.	Surface marking, anisotropy and soft-material handling.	Alloy, temper, grain direction and cosmetic surface.
Aluminum 6xxx-T6	Higher strength and common availability.	Reduced bend ductility, especially with bend line parallel to grain.	Alloy / temper, larger radius and bend orientation.

Thickness selection questions

- Does thickness come from structural need, stiffness, thread engagement, weldability, dent resistance or available stock?
- Can a return flange, rib, bead, hem or section shape create stiffness more efficiently than increasing thickness?
- Will the chosen thickness remain compatible with available laser, punch, press-brake and hardware tooling?
- Are outside dimensions allowed to grow when thickness changes, or must inside envelope remain fixed?
- Will coating, plating or anodizing change fits, threads, grounding points or cosmetic acceptance?

Avoid material callouts such as “mild steel,” “aluminum plate” or “stainless” without a grade, condition / temper and thickness.

Design note: sheet stiffness scales strongly with geometry. A small return flange or formed rib can often reduce deflection more efficiently than a modest thickness increase, but the feature adds tooling, forming and tolerance considerations.

04 CHOOSE THE FORMING PROCESS BEFORE FREEZING GEOMETRY

Choose the forming process before freezing geometry

The same angle can be produced by different processes with different tooling, force, radius and tolerance behavior.

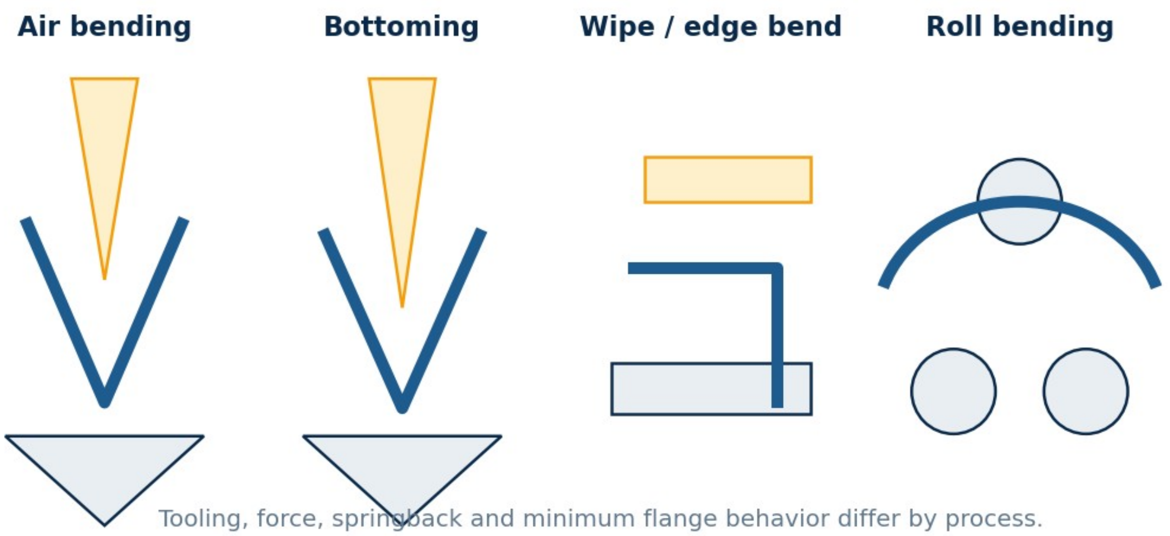


Figure 2. Common forming routes for fabricated sheet metal.

Process	Best use	Design advantage	Primary limitation
Air bending	Low-to-medium volume press-brake parts.	Flexible angles and radii with common tooling.	Springback, tool marks and minimum flange depend on V-opening.
Bottoming / coining	Higher repeatability where tooling is controlled.	Reduced angle variation in a stable setup.	Higher force and dedicated tool geometry; less flexible.
Wipe / edge bending	Flanges, panel edges and dedicated production tooling.	Good control of an edge flange.	Requires feature-specific tooling and clearance.
Roll bending	Large radii and curved panels.	Smooth large-radius forms.	End flats, radius consistency and material springback.
Stamping / forming die	High volume and complex drawn / embossed features.	High productivity and repeatability after development.	Tooling investment, development time and design freeze.

Press-brake screening checks

- The flange must sit safely on the die shoulders throughout the bend. A quick screening value is often about 0.7 x V-opening, but the actual minimum is tool-specific.
- The inside radius produced by air bending is governed mainly by material and V-opening, not only by the punch nose radius.
- Already formed flanges must clear the punch, die, ram, backgauge and machine frame during later bends.
- Long parts may require crowning, staged forming or special tooling to control angle along the length.

Do not lock a very small radius, very short flange and tight angle tolerance until the supplier confirms a tooling route.

05 BEND ALLOWANCE, NEUTRAL AXIS AND K-FACTOR

Bend allowance, neutral axis and K-factor

Flat length depends on where the neutral axis lies through the thickness during bending.

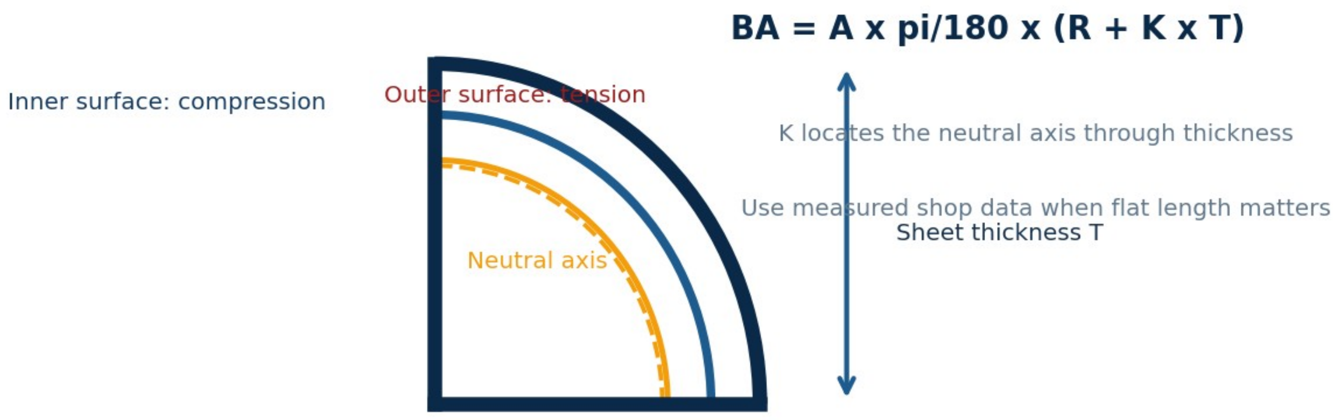


Figure 3. The neutral axis shifts toward the inside of a bend; K is the neutral-axis location divided by thickness.

Core definitions

Bend allowance BA = A x pi / 180 x (R + K x T)

Neutral-axis radius = R + K x T

K-factor = distance from inside surface to neutral axis / T

A is the angle through which the sheet is bent. Use one convention consistently across CAD, formulas and shop bend tables.

Input	Why it matters	Control method
Material / temper	Changes yield strength, anisotropy and springback.	Use the ordered grade and current material condition.
Thickness	Changes bend strain, force and developed length.	Use nominal thickness plus the shop's actual thickness strategy.
Inside radius	Changes strain and neutral-axis location.	Define a feasible radius or allow process-controlled radius.
Tooling and method	Changes radius, compression and springback.	Tie flat-pattern data to the approved press-brake setup.
K-factor / bend table	Directly changes flat length.	Derive from measured coupons or validated historical jobs.

There is no universal K-factor. CAD defaults are useful for early modeling, but production flats should come from validated material-and-tooling data.

06 BEND DEDUCTION AND FLAT-LENGTH PLANNING

Bend deduction and flat-length planning

Match the formula to the dimensioning scheme used on the drawing.

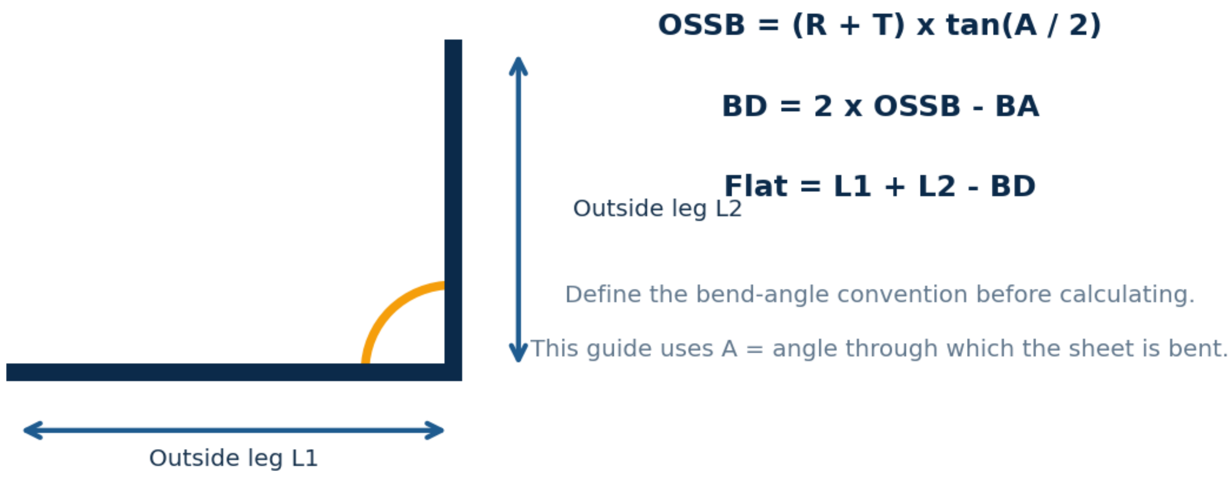


Figure 4. Outside-leg dimensions can be converted to a flat length using setback and bend deduction.

Outside-dimension method

$$\text{Outside setback OSSB} = (R + T) \times \tan(A / 2)$$

$$\text{Bend deduction BD} = 2 \times \text{OSSB} - \text{BA}$$

$$\text{Flat length} = L1 + L2 - \text{BD}$$

For multiple bends, sum straight segments and bend allowances, or subtract each bend deduction from the relevant outside dimensions.

Worked 90-degree example - screening only

Input	Value	Result
Thickness T	2.0 mm	-
Inside radius R	2.0 mm	-
K-factor K	0.40	Neutral-axis radius = 2.8 mm
Bend angle A	90 deg	BA = 4.40 mm
OSSB	(2.0 + 2.0) x tan 45 deg	4.00 mm
Bend deduction	2 x 4.00 - 4.40	3.60 mm
Outside legs	40 mm + 30 mm	Flat = 66.40 mm

A calculated flat length is not a process guarantee. Validate with a first-off coupon or measured production part before freezing a high-volume blank.

07 MINIMUM BEND RADIUS, STRAIN AND GRAIN DIRECTION

Minimum bend radius, strain and grain direction

Bendability is a material-condition problem, not only a thickness ratio.

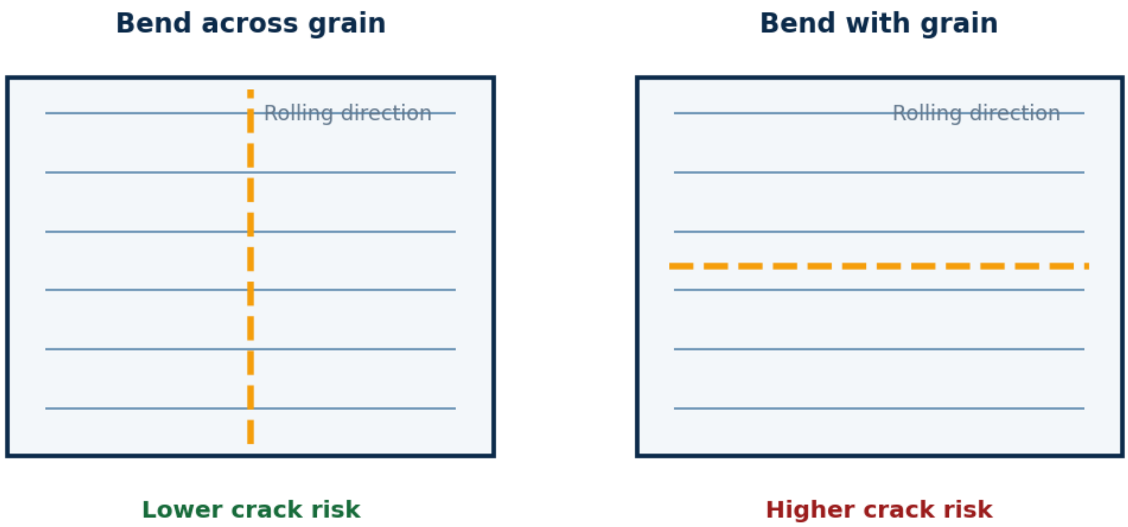


Figure 5. For anisotropic sheet, a bend line perpendicular to the rolling direction usually reduces cracking risk.

Material / condition	Practical starting range for inside radius	Risk notes
Ductile low-carbon steel	About 0.8T to 1.5T	Tighter radii may be feasible with suitable tooling; high-strength grades need more.
Austenitic stainless sheet	About 1T to 2T	Higher springback and work hardening; protect cosmetic surfaces.
Aluminum 5052-H32 / similar	About 1T to 1.5T	Verify thickness and grain direction; sharp edges can initiate cracks.
Aluminum 6061-T6 / less ductile tempers	Often 2T to 4T or more	Bend across grain where possible and validate with actual temper / thickness.
High-strength / hardened sheet	Supplier-specific	Use material data and forming trials; avoid generic “mild-steel” assumptions.

Why cracks appear

- The outer surface strain increases as the ratio R/T becomes smaller.
- Sheared or punched edges can contain burrs, fracture zones and microcracks that reduce bendability.
- Bending parallel to rolling direction can be less forgiving for directional or less-ductile material.
- Local heat, welds, work-hardened areas and protective-film contamination can change forming behavior.
- A visually acceptable bend can still have excessive thinning or dimensional springback.

The radius table above is a DFM starting point, not an acceptance standard. Confirm with the actual material certificate, thickness, bend direction and supplier tooling.

08 PLACE HOLES, SLOTS AND EDGES AWAY FROM BENDS

Place holes, slots and edges away from bends

Features near the bend zone can stretch, close, move, tear or become impossible to gauge.

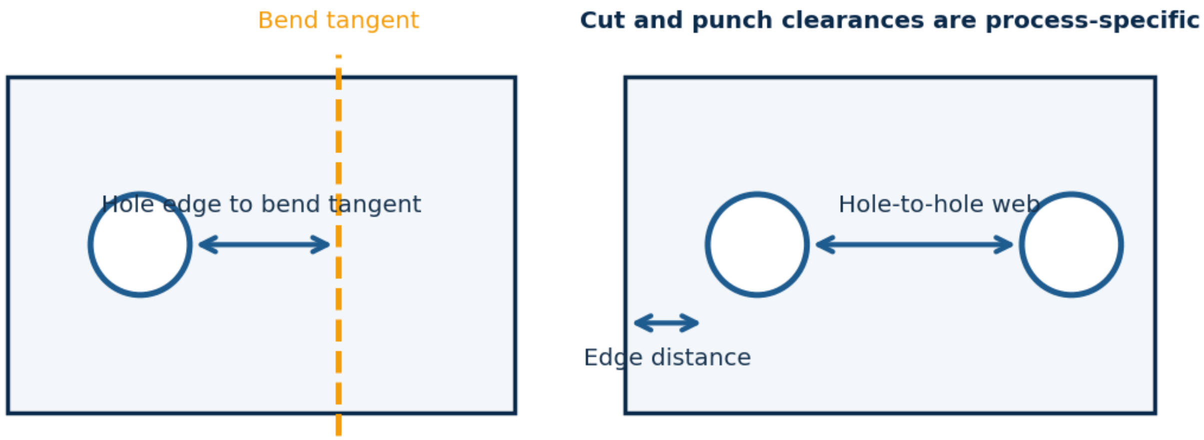


Figure 6. Distances are measured from the feature edge to the bend tangent or material edge.

Feature	Useful screening rule	Reason / action
Round hole near bend	Hole edge to tangent >= R + 2T	Reduces hole elongation and local tearing; validate for large holes and tight radii.
Slot parallel to bend	Use more clearance than a round hole	Long edges distort more easily; rotate or move the slot where possible.
Hole to sheet edge	At least about 1T for laser; punching often needs more	Prevents weak webs, breakout and edge distortion.
Punched hole diameter	About >= T for ductile steel; often >= 1.5T for stainless / hard material	Protects punch strength and hole quality; vendor rules control.
Hole-to-hole web	Common screening value >= 2T	Avoid narrow webs that distort, crack or become unstable in punching.
Countersink / dimple	Keep clear of bend strain zone and adjacent features	Formed features displace material and need tool access on both sides.

When a feature must stay close to a bend

- Add a relief slot so the bend strain does not pull through the feature.
- Punch or machine the feature after bending if position is function-critical.
- Use an obround or larger clearance hole where assembly allows.
- Add inspection of hole size and position in the final formed condition.
- Confirm whether the bend is formed toward or away from the burr side.

Do not dimension a critical hole only from a flat-pattern bend line. Dimension the functional hole position in the final formed state from stable datums.

09 BEND RELIEFS, CORNERS, FLANGES AND HEMS

Bend reliefs, corners, flanges and hems

Use local geometry to control tearing, bulging, corner overlap and unsafe edges.

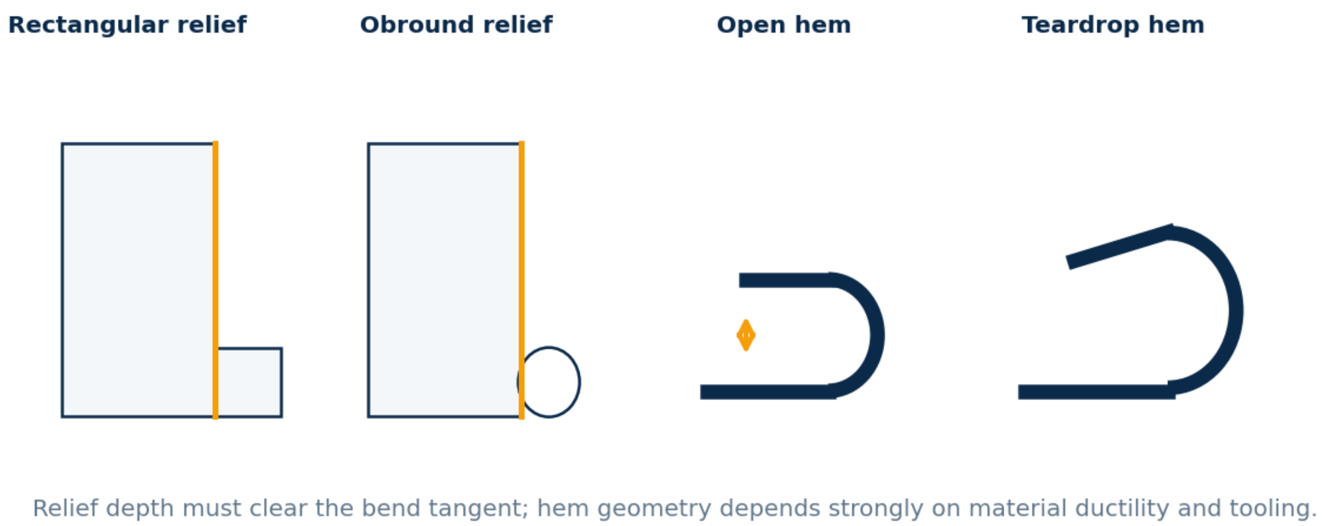


Figure 7. Common relief and hem concepts; detailed size depends on thickness, radius and tooling.

Feature	Starting design rule	Common failure
Bend relief width	At least about T; larger for thick or less-ductile sheet.	Relief closes, tears or leaves a sharp notch.
Bend relief depth	Extend past the bend tangent, typically at least R + T plus process margin.	Corner bulges or tears before the bend completes.
Inside box corner	Provide overlap, relief or controlled corner gap.	Material collision, split corners or inconsistent weld gap.
Minimum flange	Must remain supported on the die; screen near 0.7 x V-opening.	Flange slips into die or angle varies.
Open hem	Keep a controlled gap and specify its functional purpose.	Gap closes unevenly or traps coating / contamination.
Closed / flattened hem	Use only with suitable ductility and tooling.	Cracking, spring-open, waviness and doubled-thickness stack-up.

Corner design choices

- Open corner: simple and low stress, but may need a cosmetic or sealing solution.
- Overlap corner: easy to locate and weld, but creates double thickness and visible edge steps.
- Mitered corner: clean appearance, but sensitive to flat-pattern accuracy and bend sequence.
- Tab-and-slot corner: improves self-location and fixture reduction; include insertion clearance and weld access.
- Hemmed edge: improves safety and stiffness but adds forming steps and thickness stack-up.

Reliefs should solve a defined forming problem. Oversized reliefs may create appearance, sealing, strength or sharp-edge problems of their own.

10 LASER CUTTING AND THERMAL-CUT EDGE DESIGN

Laser cutting and thermal-cut edge design

Cut capability is not the same as guaranteed feature tolerance or edge acceptance.

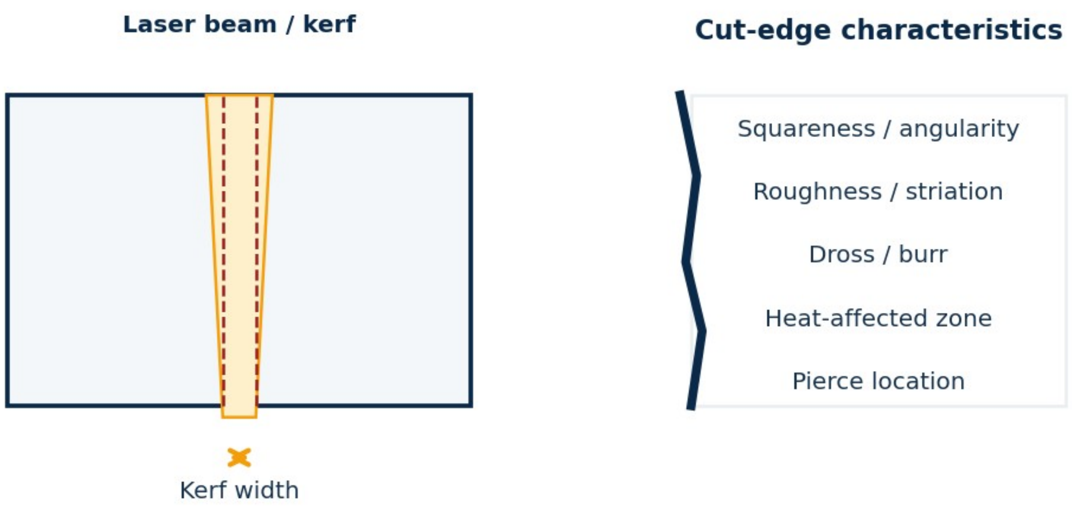


Figure 8. Kerf and cut-edge quality vary with material, thickness, assist gas, focus, speed and heat accumulation.

Design topic	Guidance	Drawing / inspection action
Kerf compensation	Normally handled by machine programming, not by changing nominal part geometry.	Dimension the finished feature; do not add “kerf allowance” to the drawing.
Small holes	Laser can cut below thickness in some materials, but roundness, taper and dross can degrade.	Define critical hole function and allow drilling / reaming if needed.
Dense patterns	Heat accumulation can distort thin sheets and narrow webs.	Increase spacing, split operations or use micro-joints strategically.
Pierce marks	Piercing may create a local mark, spatter or lead-in witness.	Keep pierce / lead-in away from cosmetic or sealing edges.
Cut edge	May include angularity, striation, burr, dross and HAZ.	Specify an ISO 9013 class or explicit acceptance only when needed.
Protective film	Some films are laser-compatible; others melt, smoke or contaminate.	State film requirement and confirm compatibility with the supplier.

ISO 9013 scope note

ISO 9013:2017, amended in 2024, classifies geometrical quality of thermal cuts. Its published scope includes laser cuts from 0.5 mm to 32 mm. It does not automatically impose a quality class unless the drawing or contract invokes one.

Tight profile tolerance on a long thin blank may be limited by residual stress and heat distortion even when the machine’s positioning accuracy is much better.

11 PUNCHING, BLANKING AND FORMED FEATURES

Punching, blanking and formed features

Punch quality and minimum geometry depend on material, clearance, tool strength and feature direction.

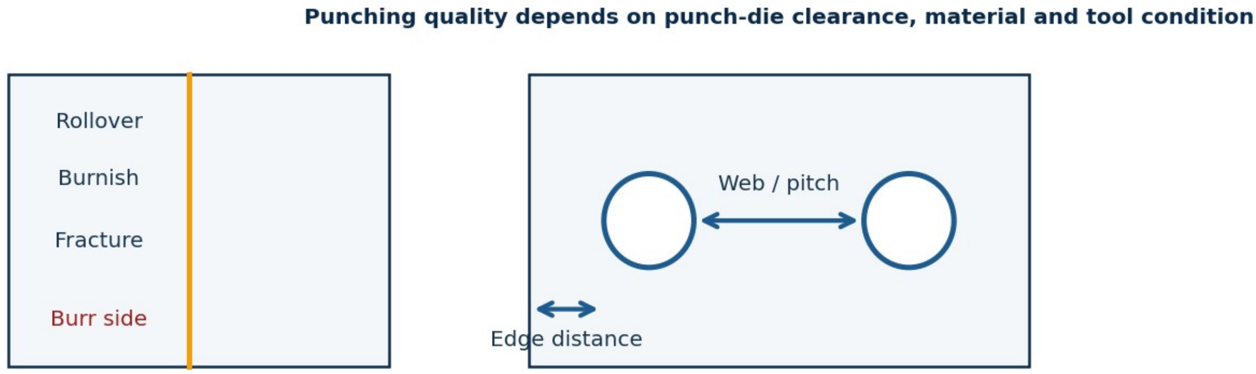


Figure 9. A punched edge normally contains rollover, burnish, fracture and a burr on the die side.

Feature / issue	Design guidance	Control point
Burr direction	Choose the burr side relative to assembly, sealing, coating and user contact.	Show burr-side requirement only when functionally important.
Minimum hole / slot width	Tool vendors often start near T for ductile steel and larger for stainless / hard material.	Confirm punch strength, clearance and acceptable edge quality.
Narrow webs	Avoid webs that are too narrow to remain stable during punching.	Increase web, change sequence or use laser cutting.
Nibbling	Useful for large openings without dedicated tooling.	Accept witness marks or require secondary finishing.
Louvers / embosses / beads	Require material flow, forming height and access on both sides.	Define orientation, height and cosmetic side.
Countersinks / extrusions	Material thickness limits useful depth and thread engagement.	Use formed-feature dimensions and inspect in final condition.

Blanking process choice

Volume / geometry	Preferred starting route	Why
Prototype / low volume, complex outline	Laser cutting	No hard tooling; rapid revision.
Repeated holes / standard forms	CNC turret punching	Fast recurring features; standard forming tools.
High-volume stable design	Progressive / transfer / compound die	Lowest cycle time after tooling investment.
Thick or very large plate-like sheet	Laser / plasma / waterjet as appropriate	Capacity and edge-quality trade-offs.

A punched hole is not a machined bore. For bearing fits, precision dowels, seals or critical threads, plan a secondary sizing operation.

12 HARDWARE INSERTION AND THREADED FEATURES

Hardware insertion and threaded features

Select the joining method by access, sheet thickness, load, finish and service requirements.

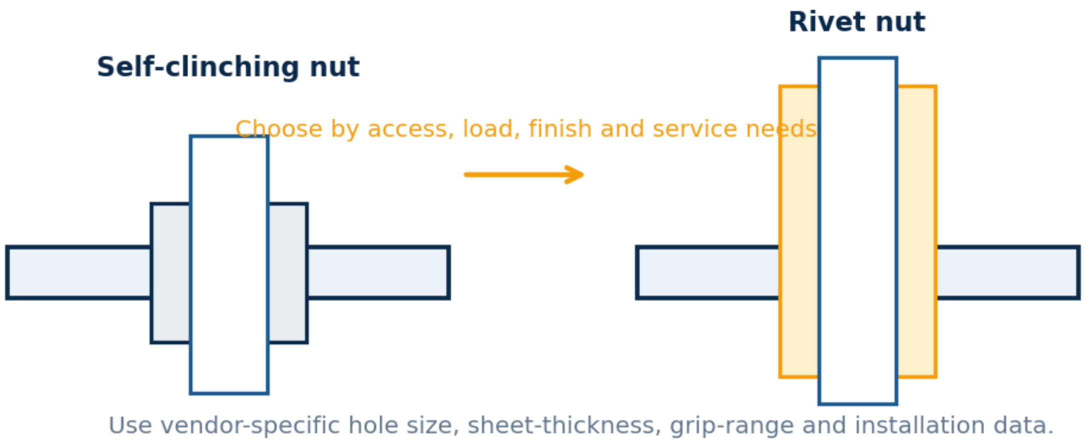


Figure 10. Self-clinching and rivet-nut systems use different installation access and sheet-engagement principles.

Method	Strengths	Design checks
Self-clinching nut / stud / standoff	High productivity, flush options and repeatable location.	Vendor hole size, minimum sheet thickness / hardness, edge distance and installation access.
Rivet nut	One-side installation and useful for closed sections.	Grip range, anti-rotation, pull-out, head style and installation tool access.
Tapped extrusion	Creates more thread engagement from thin sheet.	Material ductility, extrusion height, crack risk and tool access.
Direct tapping	Simple for thicker sheet or low-load service.	Minimum engagement, burr, coating and thread class.
Weld nut / stud	High load and common in welded assemblies.	Weld access, spatter, distortion, positional tolerance and finish sequence.
Loose fastener + clearance hole	Easy sourcing and service replacement.	Two-side access, anti-rotation and assembly tool clearance.

Sequence rules

1. Cut and deburr the specified hole; do not use an approximate drill size for proprietary hardware.
2. Form bends in an order that protects installed hardware and leaves installation-tool access.
3. Install self-clinching hardware before finish unless the approved fastener and process require otherwise.
4. Mask threads, grounding surfaces and press-fit interfaces during coating.
5. Inspect presence, orientation, flushness, thread function and push-out / torque resistance where risk requires.

Do not place hardware so close to a bend that insertion deforms the flange or later bending collides with the fastener.

13 MULTI-BEND PARTS, BOXES AND PRESS-BRAKE ACCESS

Multi-bend parts, boxes and press-brake access

A part can be dimensionally valid in CAD but impossible to form in the proposed sequence.

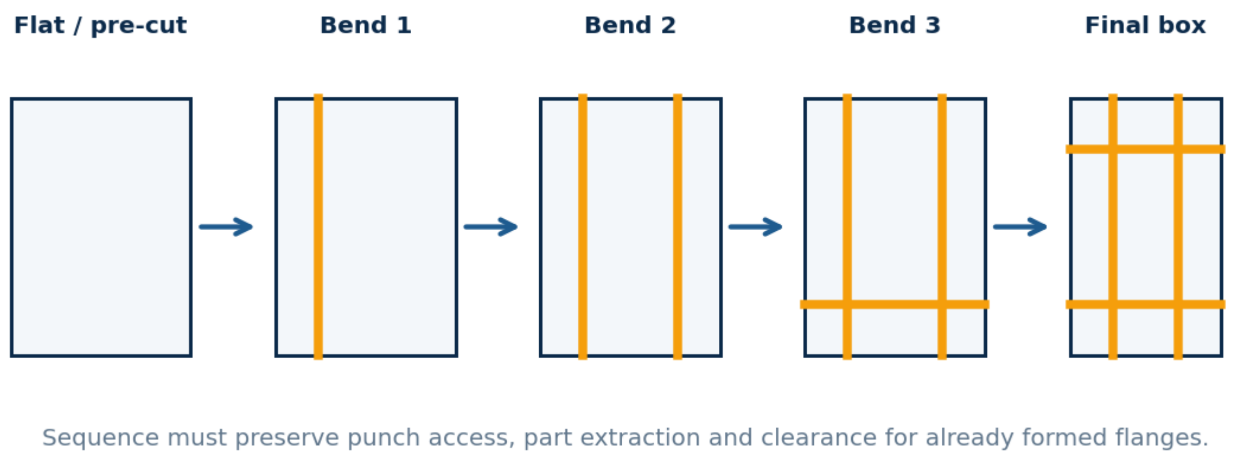


Figure 11. Bend sequence must preserve tooling access and part extraction.

Access question	Failure mode if ignored	Design response
Can the flange reach the bend line without hitting the punch or ram?	Part collision before the bend completes.	Use gooseneck tooling, change sequence, add relief or redesign flange.
Can the formed part be removed from the tooling?	Part becomes trapped after a closing bend.	Open a corner, split the part or use staged / special tooling.
Can the backgauge locate each bend?	Unstable or manual location causes angle / length variation.	Add gaugeable edges, tabs or temporary extensions.
Will prior bends fit between die shoulders and machine frame?	Flange or box wall interferes with the press brake.	Check 3D tool envelope, not only part-to-part clearance.
Are inside box dimensions controlled across multiple bends?	Tolerance stack-up closes the box or shifts holes.	Define functional datums and allow process-appropriate profile tolerance.

Box and channel design practices

- Keep repeated bends on common tooling and, where possible, use one inside radius family.
- Provide corner relief that prevents material overlap but does not expose an unsafe edge.
- Avoid unnecessary closed dimensions that over-constrain flange lengths and overall size simultaneously.
- For deep boxes, consider split construction, welding, clinching or dedicated forming rather than forcing press-brake geometry.
- Use tab-and-slot or locating features carefully: include cutting clearance and account for coating thickness.

Request a supplier bend-feasibility review whenever the part has deep returns, short internal flanges, multiple closing bends or hardware near the tool path.

14 WELDED SHEET METAL ASSEMBLIES AND DISTORTION CONTROL

Welded sheet metal assemblies and distortion control

Thin sheet responds strongly to local heat input, weld shrinkage and poor restraint.

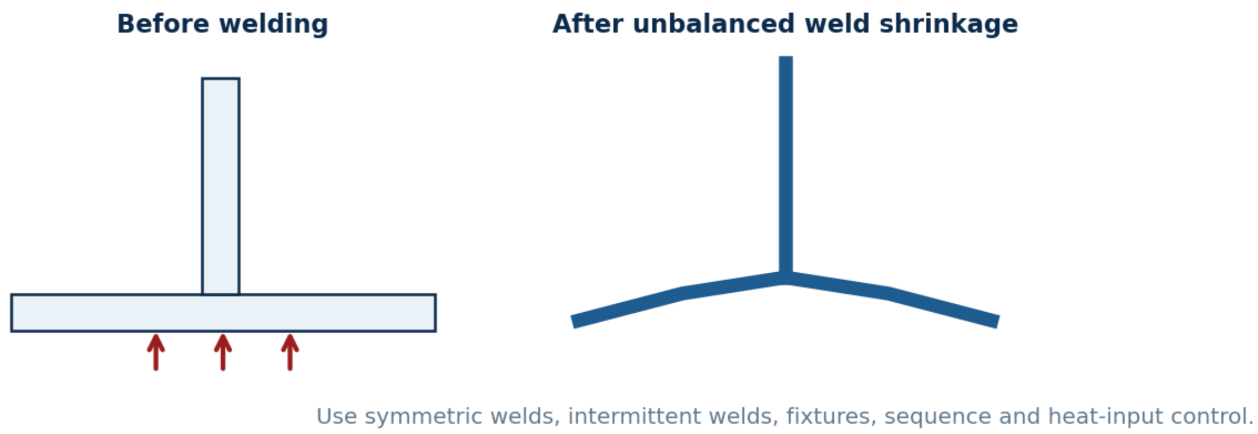


Figure 12. Unbalanced weld shrinkage can bow panels and rotate flanges.

Design decision	Preferred practice	Why
Joint type	Use lap, flange, tab-slot or controlled butt joints that support repeatable fit-up.	Reduces gaps, fixture dependence and burn-through risk.
Weld length	Use the minimum weld needed for load, sealing and durability.	Reduces heat, cycle time and distortion.
Weld balance	Distribute welds symmetrically around the neutral region where possible.	Balances shrinkage and angular distortion.
Weld sequence	Use skip, back-step or alternating sequence based on the qualified process.	Avoids concentrating heat in one area.
Cosmetic surface	Keep welds, spatter and grinding away from class-A faces where possible.	Grinding can thin sheet and create visible waviness after finish.
Datum strategy	Locate on robust formed features, not flexible unsupported panels.	Improves assembly repeatability and inspection.

Design controls for thin sheet

- Add return flanges, beads or temporary strongbacks to stabilize large panels.
- Use tabs and slots for location, but provide clearance for cut tolerance and coating.
- Avoid specifying continuous welds when intermittent welds satisfy strength and sealing needs.
- Specify weld size, length, spacing and quality level; do not rely on “weld all around” by habit.
- Plan post-weld straightening only when necessary and identify dimensions that must be checked after welding.

AWS D9.1/D9.1M:2018 covers arc and braze welding requirements for nonstructural sheet metal fabrications. Structural or safety-critical work may require a different governing code.

15 DRAWINGS, TOLERANCES, SURFACE FINISH AND INSPECTION

Drawings, tolerances, surface finish and inspection

Control the final functional geometry without turning process information into conflicting requirements.

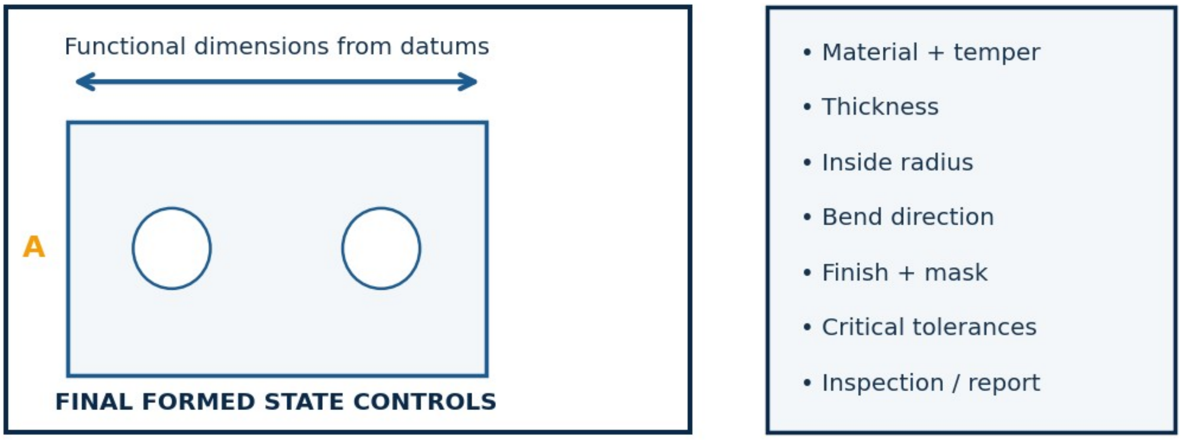


Figure 13. A manufacturable drawing identifies material, formed-state datums, bend intent, finish and inspection needs.

Drawing item	Minimum information	Common mistake
Material	Standard / grade, temper or condition, thickness and allowed substitution.	Using a generic family name only.
Bends	Inside radius or permitted range, bend direction and critical angles.	Dimensioning punch radius instead of required part geometry.
Dimensions	Functional dimensions from stable datums in the final state.	Duplicating flat and formed dimensions that can conflict.
General tolerance	A declared standard or explicit block tied to the manufacturing state.	Applying one tight tolerance to every laser and formed feature.
Edges / burrs	Defined edge-break, burr direction or ISO 13715 indication when functional.	Writing “deburr all edges” without an acceptance limit.
Finish	System, color, gloss / texture, thickness, masked areas and cosmetic class.	Ignoring coating buildup in holes, threads and fits.
Inspection	Critical characteristics, sampling, reports and measurement datum.	Requesting full reports without defining what must be measured.

Tolerance strategy

- Use profile or position controls when they express function better than many chained linear dimensions.
- Separate flat blank tolerances from final formed tolerances; the final part usually controls assembly function.
- Use realistic angle tolerances based on flange length, material, tooling and measurement method.
- Allow wider tolerances on nonfunctional cut edges, hidden reliefs and weld gaps where performance is unaffected.
- Identify class-A surfaces and viewing conditions instead of applying cosmetic requirements to every face.

ISO 129-1 covers dimension presentation; ISO 13715 covers undefined edges; ASME Y14.5 or ISO 1101 can define geometric tolerances. State the adopted system clearly.

16 SHEET METAL DESIGN REVIEW CHECKLIST

Sheet metal design review checklist

Use the checklist before RFQ release and again before production drawing approval.

No.	Release check	Category
1	Material standard, grade / temper and thickness are explicit.	Material
2	Rolling / grain direction is defined where bendability or appearance depends on it.	Material
3	Surface condition, protective film and cosmetic faces are identified.	Material
4	Blanking route is feasible for thickness, holes, webs and edge quality.	Cutting
5	Cut-edge, burr and edge-break requirements are measurable.	Cutting
6	Inside radii are compatible with material and available tooling.	Bending
7	Minimum flange lengths are compatible with the expected V-opening.	Bending
8	Bend sequence, punch clearance and part extraction are feasible.	Bending
9	Critical holes and slots are clear of bend strain zones.	Features
10	Reliefs clear the bend tangent and do not create unsafe edges.	Features
11	Hems, embosses, louvers and extrusions have suitable material flow and access.	Features
12	Hardware hole sizes, sheet limits and installation access follow vendor data.	Hardware
13	Hardware, threads and grounding points are protected from finish buildup.	Hardware
14	Tabs, slots and locating features include assembly and coating clearance.	Assembly
15	Weld length and size are no greater than function requires.	Welding
16	Weld sequence, fixture strategy and post-weld inspection are defined where needed.	Welding
17	Functional datums control the final formed / welded state.	Drawing
18	No duplicated dimensions over-constrain the flat and formed geometry.	Drawing
19	General and critical tolerances are process-appropriate.	Drawing
20	Finish system, color, texture, thickness and masked areas are complete.	Finish
21	Packaging protects cosmetic faces, edges, threads and corrosion-sensitive areas.	Packaging
22	Inspection method can access and measure every critical characteristic.	Inspection
23	First-article and report requirements match project risk.	Inspection
24	Expected annual volume supports the selected process and tooling investment.	Commercial
25	Supplier review is required for any unresolved red-flag geometry.	Release

Release rule: unresolved material, bend-access, hardware, tolerance or inspection risks should be recorded as open actions - not hidden in a generic drawing note.

17 STANDARDS, TECHNICAL SOURCES AND RFQ HANDOFF

Standards, technical sources and RFQ handoff

Use current controlled editions and verify customer-specific requirements before release.

ISO 7438:2020 - Metallic materials - Bend test. Official source

ISO 9013:2017 + Amd 1:2024 - Thermal cutting - classification, geometrical specifications and quality tolerances. Official source

ISO 13715:2017 - Technical product documentation - edges of undefined shape. Official source

ISO 129-1:2018 + Amd 1:2020 - General principles for presentation of dimensions and tolerances. Official source

ASME Y14.5-2018 (R2024) - Dimensioning and tolerancing / GD&T. Official source

AWS D9.1/D9.1M:2018 - Sheet Metal Welding Code for nonstructural sheet metal fabrications. Official source

ASTM A1008/A1008M-26 - Cold-rolled carbon, structural and high-strength steel sheet. Official source

ASTM B209/B209M-21a - Aluminum and aluminum-alloy sheet and plate; verify current edition at RFQ. Official source

Standards define requirements and notation; they do not replace a process-specific DFM review, shop bend data, tool clearance check or first-off validation.

Minimum RFQ handoff

File / information	Required content
2D drawing	Material, thickness, finish, critical dimensions, tolerances, hardware, welds and revision.
3D model	Final formed geometry; neutral format such as STEP when requested.
Flat pattern	Optional / reference unless the customer owns and controls the developed blank.
Volume and schedule	Prototype quantity, batch quantity, annual demand and target delivery.
Quality requirements	FAI, inspection report, certificates, cosmetic standard and critical characteristics.
Packaging / logistics	Destination, shipment mode, corrosion period, labeling and pack quantity.

Contact KingsForm Metalworks

Engineering RFQ: info@kingsformmetalworks.com
www.kingsformmetalworks.com