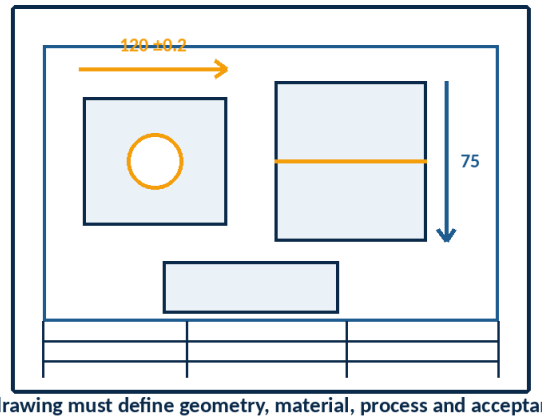
KINGSFORM METALWORKS

ENGINEERING TOOLKIT

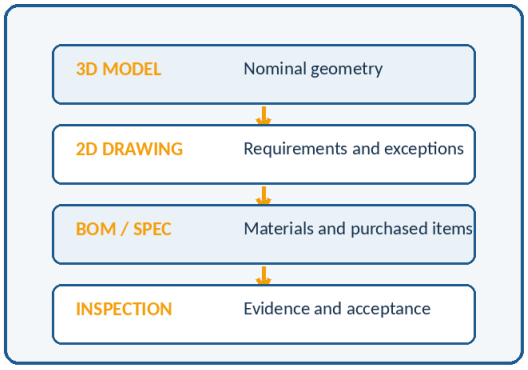
MANUFACTURING DRAWING GUIDE

Views, dimensions, datums, materials, processes, tolerances, revision control and inspection handoff

MANUFACTURING DRAWING



A drawing must define geometry, material, process and acceptance.



DOCUMENT	VERSION	RELEASE DATE	LANGUAGE
KFR-011	1.0	2026-07-17	English

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

Engineering reference only. The released drawing, model, contract, governing specification and customer requirements remain controlling.

01 SCOPE AND USE

Scope and how to use this guide

A practical reference for preparing fabrication drawings that can be quoted, manufactured, inspected and revised without hidden assumptions.

Use this guide before an RFQ or release package is issued to a metal-fabrication supplier.

The guide covers 2D drawings used with or without a 3D model. It focuses on the information needed for fabricated metal parts and assemblies: drawing hierarchy, views, datum strategy, dimensions, holes and threads, tube bends, sheet-metal forming, weldments, materials, surface finish, tolerances, key characteristics, revisions and supplier handoff.

Recommended use

1. Identify the governing drawing system and standards before creating views or notes.
2. Define functional interfaces and datums before dimensioning non-functional geometry.
3. Specify material, product form, condition and special processes as controlled requirements.
4. Review manufacturability, measurement access and reporting before release.
5. Issue one coordinated revision across the model, drawing, BOM, specifications and inspection plan.

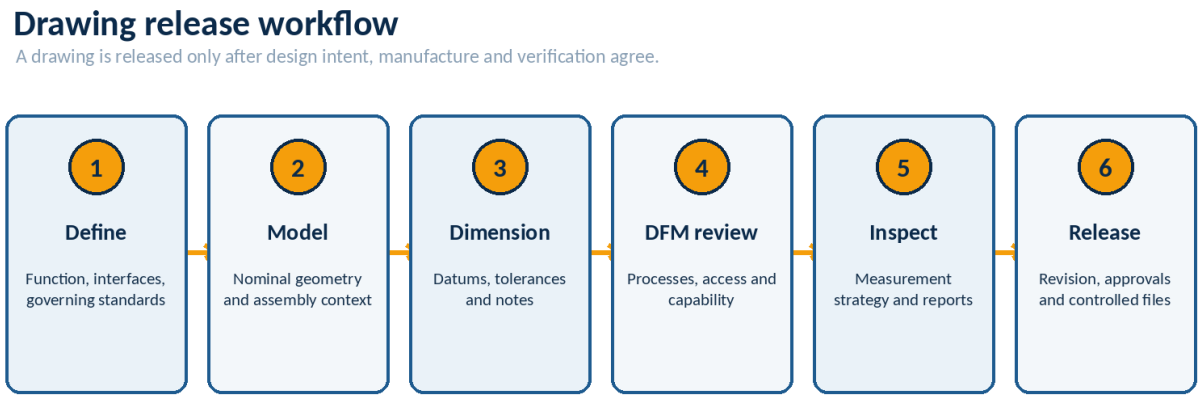
Contents

Primary audience: designers, purchasing engineers, quality teams and suppliers reviewing drawing completeness before production.

02 RELEASE WORKFLOW

Drawing release workflow

The drawing is a controlled engineering output, not a late-stage annotation exercise.



Freeze the requirement source before adding dimensions; freeze inspection before final release.

Figure 1. Requirement-driven sequence for drawing creation and release.

Gate	Questions to answer	Release evidence
Requirement source	Which contract, standard, interface and performance requirement controls?	Approved requirement list / interface definition
Geometry	Does the model represent the intended nominal shape and assembly context?	Reviewed native model and configuration
Definition	Are all size, location, material, process and acceptance requirements explicit?	Completed drawing and notes
Manufacturability	Can the supplier make the part with available processes and inspection access?	DFM review / open-issue closure
Control	Do all released files carry the same part number, revision and approval state?	Released package and revision record

A clean drawing has no “tribal knowledge” requirement that exists only in email, chat or a previous sample.

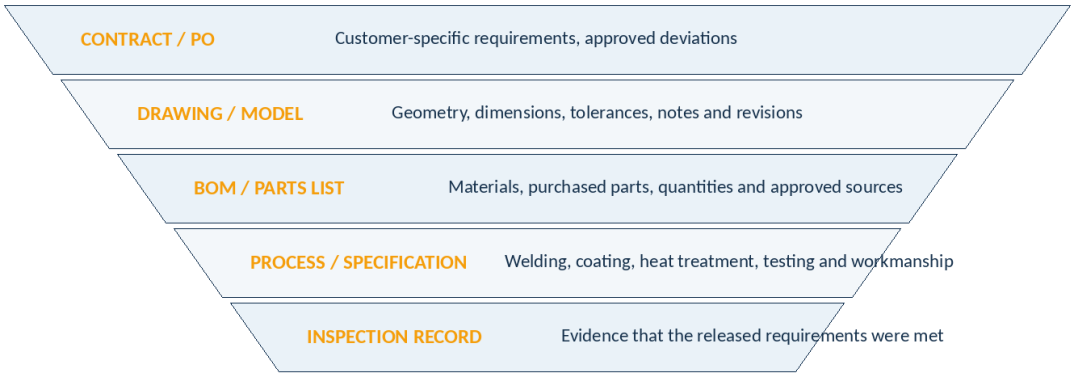
03 DOCUMENT ARCHITECTURE

Product-definition package and precedence

Separate geometry, purchased-item data, process specifications and inspection evidence, then state which document controls.

Product-definition package

Use a clear hierarchy so suppliers know which document controls each requirement.



State precedence explicitly when a 3D model, drawing, specification and customer standard may conflict.

Figure 2. Typical hierarchy for a controlled manufacturing package.

Document	Minimum content	Common risk
Part drawing	Part number, revision, material, dimensions, tolerances, finish and notes.	Geometry is clear but material condition or post-process size is undefined.
Assembly drawing	Item identity, quantity, location, joining, interfaces and functional checks.	BOM and balloons do not match the visible assembly.
BOM / parts list	Item number, part number, revision or approved configuration and quantity.	Supplier substitutes a visually similar but functionally different item.
Process specification	Detailed welding, coating, heat-treatment, cleaning or testing method.	A generic note cites a standard without class, level or acceptance criteria.
3D model / digital definition	Nominal geometry, configurations, model-based requirements if adopted.	The model and 2D drawing represent different revisions.
When both model and drawing are supplied, state whether the drawing is fully dimensioned, reduced-dimension, or model-based.		

Views, sections and details

Use a minimal but complete view set that exposes manufacturing and inspection features.

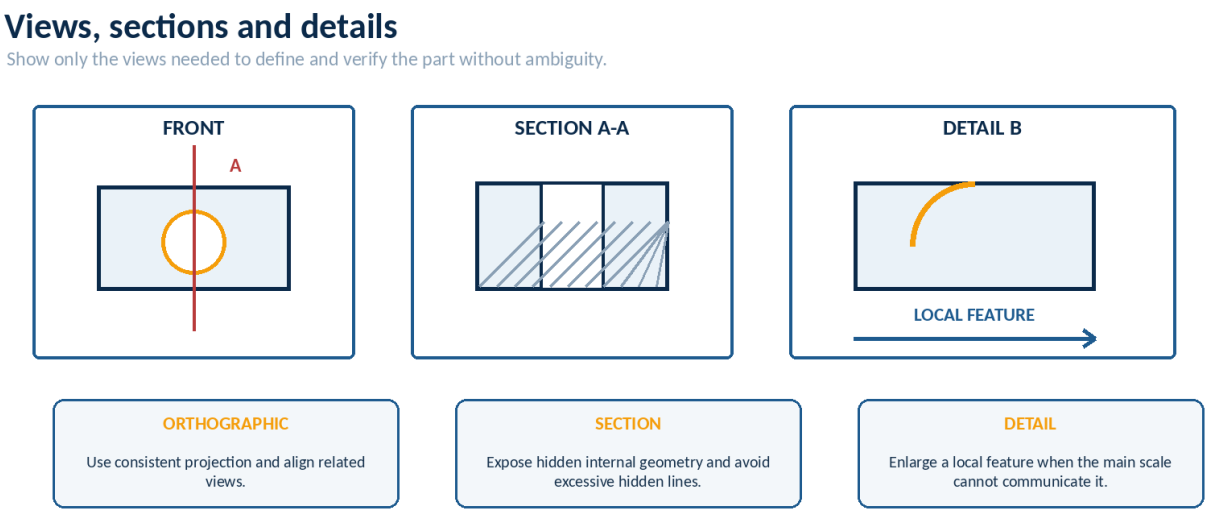


Figure 3. Orthographic, section and detail views serve different communication purposes.

View practice	Use when	Avoid
Principal orthographic views	The visible outline and feature relationships can be defined directly.	Adding redundant views that repeat dimensions and create update risk.
Section / cut	Internal geometry, wall thickness, weld preparation or hidden interfaces matter.	Dense hidden lines that are difficult to interpret or inspect.
Detail view	A local edge, groove, thread, relief or small feature needs a larger scale.	Changing the local geometry in the detail without matching the parent view.
Auxiliary view	An inclined surface must be shown true size.	Dimensioning an inclined feature in a foreshortened projection.
Exploded / assembly view	Item identity and assembly order need clarification.	Using an exploded view as the only definition of final position.

- Identify the projection method and keep related views aligned.
- Use centerlines, cutting-plane lines and leader lines consistently with the selected drawing standard.
- Do not dimension to hidden lines when a section or additional view can show the feature directly.

Datum and dimension strategy

A good drawing makes the assembly and inspection coordinate system explicit.

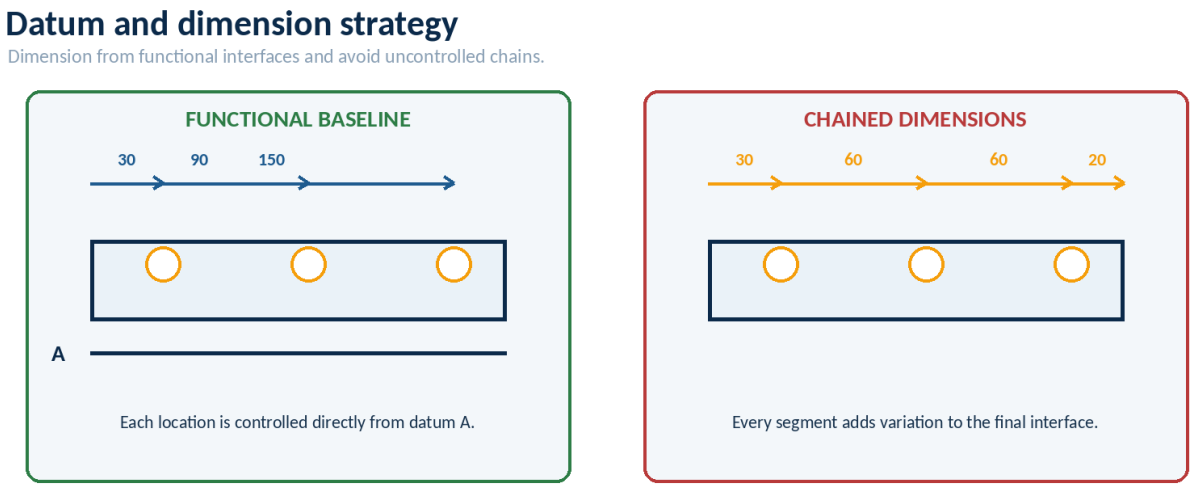


Figure 4. Baseline dimensioning controls functional locations more directly than a closed chain.

Rule	Engineering reason	Typical application
Select datums from functional interfaces.	Reduces error between the part and the feature that actually locates or loads it.	Mounting face, bearing seat, mating edge, tube end or fixture pad.
Use baseline / ordinate dimensions for related locations.	Prevents tolerance accumulation across a chain of intermediate features.	Hole patterns, repeated slots and frame attachment points.
Do not duplicate a controlled dimension.	Two values for one relationship can conflict after a revision.	Model dimension plus drawing note, or two orthographic views.
Use reference dimensions only as information.	Prevents calculated or redundant values from becoming acceptance criteria.	Overall envelope derived from controlled feature dimensions.
Control the result of a stack, not every part equally.	Allows economical tolerances where they matter most.	Door gap, shaft alignment, bracket position and assembly flushness.

A dimension should answer one question: what feature is controlled, from which datum or origin, and with what acceptance zone?

06 TOLERANCE NOTATION

Dimension types and tolerance logic

Choose the notation that communicates the permitted variation and inspection decision clearly.

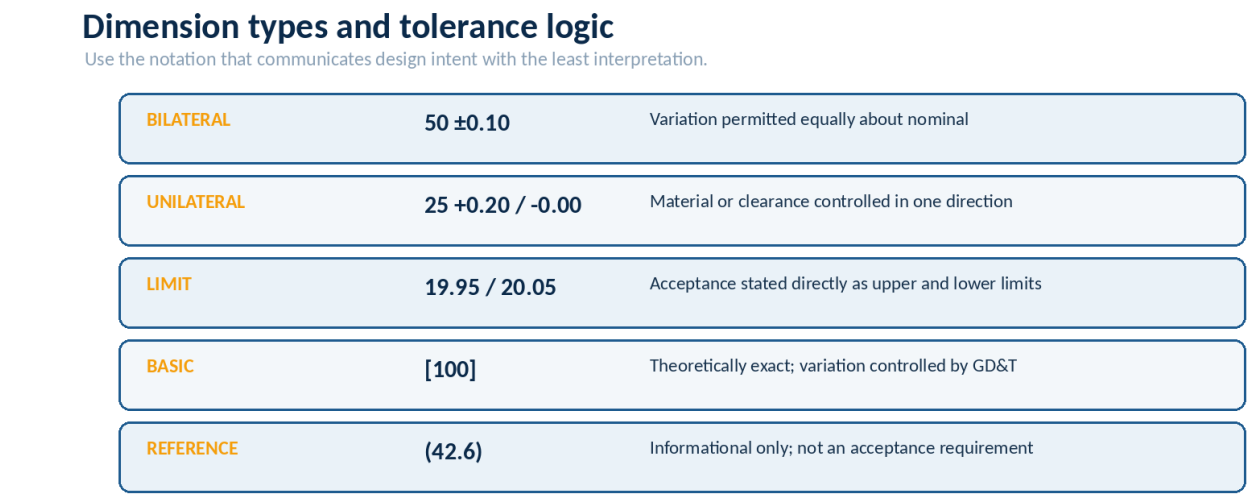


Figure 5. Common linear-dimension presentations and their intended use.

Control	Best use	Design note
Bilateral tolerance	A centered process and symmetric functional allowance.	Do not assume the process will naturally center on nominal.
Unilateral tolerance	Clearance, material boundary or one-sided functional limit.	Useful for minimum wall, maximum envelope and minimum clearance.
Limit dimensions	Direct upper and lower acceptance limits.	Avoid ambiguity about sign or rounding near the limits.
Basic dimension	Theoretically exact location, orientation or profile controlled by GD&T.	A basic dimension does not carry a plus/minus tolerance.
Reference dimension	Convenience information calculated from controlled data.	Show in parentheses or by the convention required by the adopted standard.

- Use decimal precision to support the stated tolerance; extra zeros do not create process capability.
- State units and decimal-place rules in the title block or general notes.
- Avoid a closed dimension loop unless one value is clearly reference or the closure condition is intentionally controlled.

Holes, slots, threads, countersinks and counterbores

Callouts should define the feature in its final usable condition.

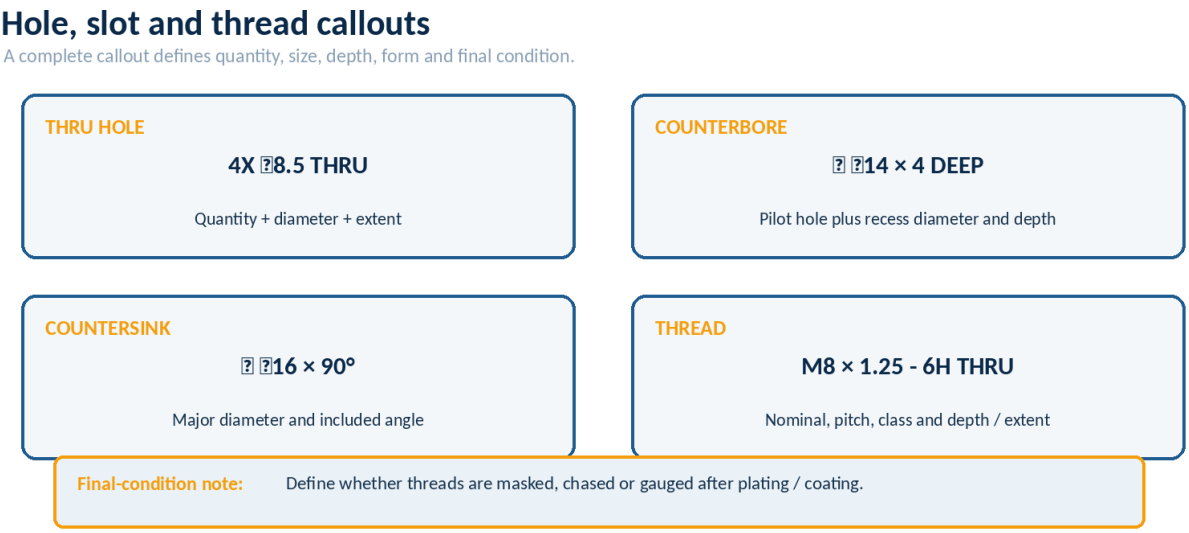


Figure 6. A complete feature callout includes quantity, form, size, extent and final condition.

Feature	Information to include	Manufacturing / inspection concern
Clearance hole	Quantity, diameter, through / depth, positional control and deburr requirement.	Laser, punch and drill processes produce different edge and size capability.
Slot	Width, end radius or full geometry, orientation, length and positional control.	A slot may be for adjustment or location; the tolerance strategy differs.
Tapped hole	Thread system, pitch, class, depth / through, minimum full thread and post-finish condition.	Blind holes need allowance for drill point, incomplete thread and debris.
Counterbore / spotface	Pilot feature, recess diameter, depth and surface requirement.	The fastener head and washer must seat without interference.
Countersink	Pilot feature, major diameter or depth, included angle and side.	Coating and edge break can alter seating and flushness.
Specify whether plated or coated threads are masked, chased, tapped after finish, or gauged in the final condition.		

Tube and bent-part drawing requirements

A tube drawing must define the stock, bend path, clocking and end features in a measurable coordinate system.

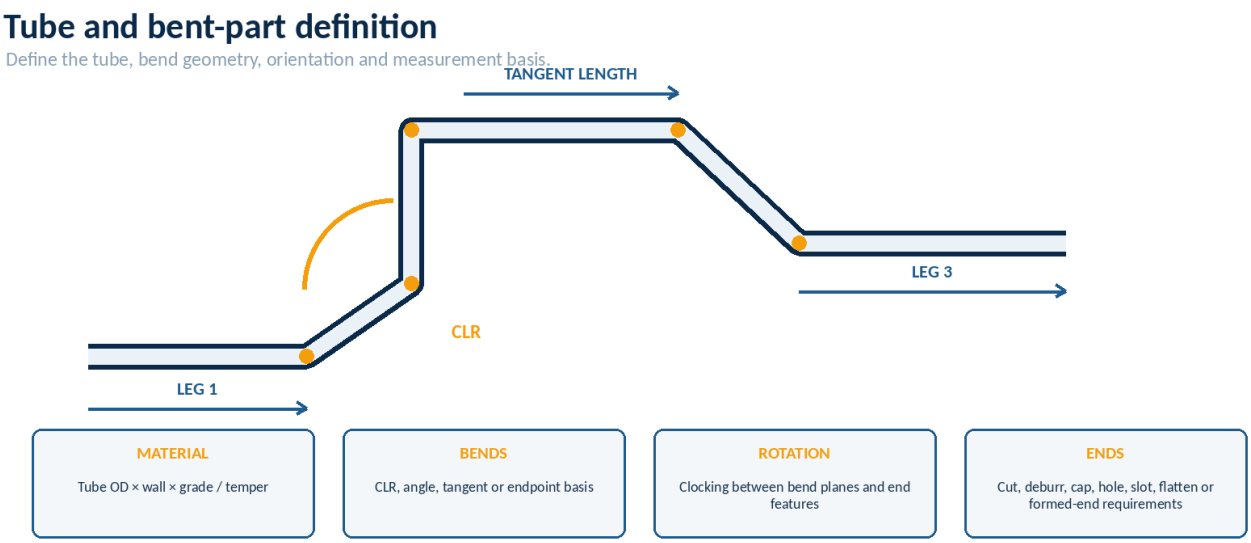


Figure 7. Bent-tube definition requires more than overall length and bend angle.

Requirement	Minimum drawing content	Why it matters
Stock definition	OD or section size, wall thickness, grade, condition and welded / seamless requirement.	The same nominal size can have different formability, strength and seam behavior.
Bend geometry	Centerline radius, angle, tangent points or endpoint coordinates and bend plane.	Avoids uncertainty between inside, outside and centerline measurement.
Clocking	Rotation between bend planes and orientation of holes, seams or end fittings.	Small angular errors can create large endpoint errors.
End condition	Cut length, squareness, chamfer, deburr, cap, flatten, swage, hole or slot.	End features often control assembly and inspection.
Acceptance	Endpoint position, angle, rotation, ovality, wall thinning, surface and gauge method.	A profile gauge may be more practical than many independent dimensions.

For multi-bend parts, supply a bend table or LRA / YBC data only when the coordinate convention and revision are controlled.

Sheet-metal drawing requirements

The drawing should control the finished formed part while giving the supplier enough process information to create a stable flat pattern.

Sheet-metal drawing package

The formed state, flat pattern and hardware plan must reference one controlled manufacturing definition.

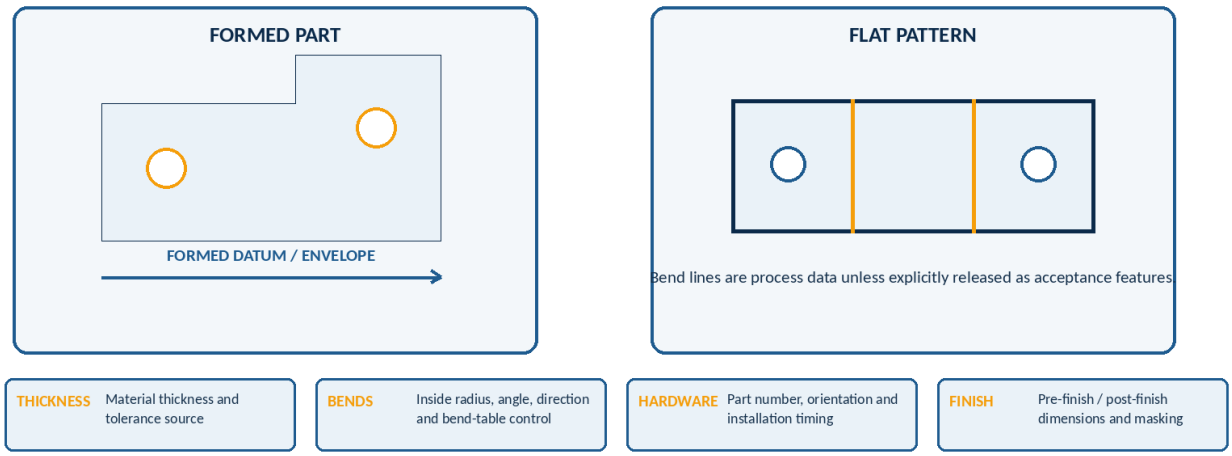


Figure 8. Formed geometry and flat-pattern process data must remain coordinated.

Topic	Drawing requirement	Common error
Material / thickness	Grade, temper or condition, thickness and thickness-tolerance source.	Using “mild steel” or “aluminum” without a grade and form.
Bends	Inside radius, angle, direction, critical flange dimensions and any controlled bend sequence.	Dimensioning both inside and outside geometry redundantly.
Flat pattern	State whether the flat is supplier-developed or a released acceptance document.	Treating a CAD flat pattern as exact when the bend table is unvalidated.
Reliefs / corners	Relief type, corner condition, minimum flange and edge requirements.	Cracks, overlap or tearing at box corners and short flanges.
Hardware	Exact insert / stud / nut part number, orientation and installation timing.	Pressing hardware into material or finish outside the supplier specification.
Finish	Masking, contact areas and whether dimensions apply before or after coating.	Holes or mating gaps close after powder coat or plating.

- Control functional flange position from formed datums, not only from flat-pattern bend lines.
- Show grain direction when cracking, appearance or stiffness depends on rolling direction.

Weldment drawing requirements

Define the joint, weld, quality level and final geometry as one coordinated requirement set.

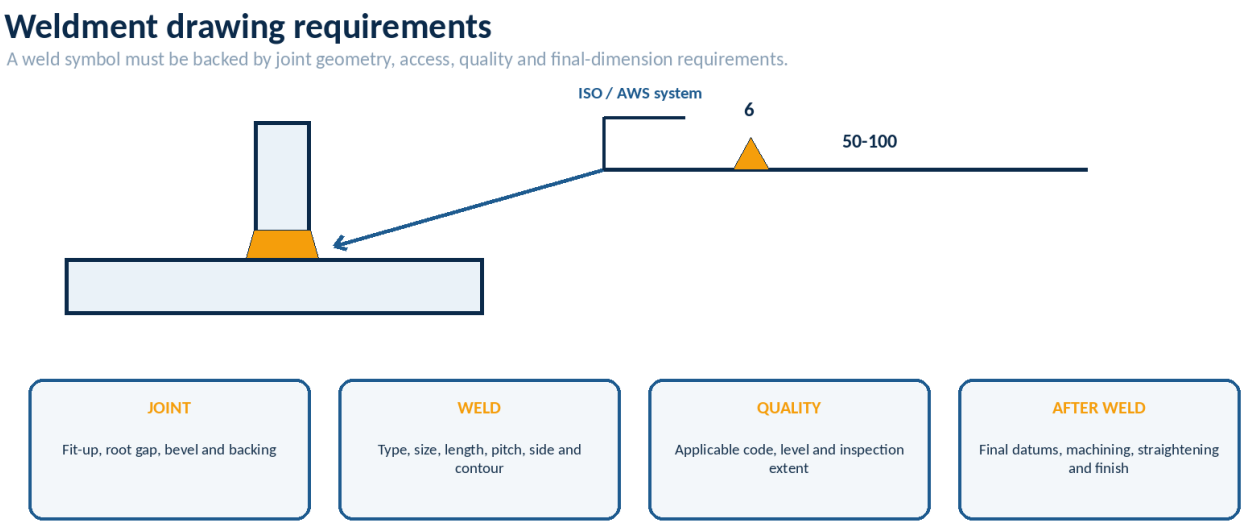


Figure 9. The symbol identifies the weld; joint geometry and acceptance still need to be defined.

Drawing item	Required definition	Practical note
Joint preparation	Root opening, bevel angle, land, backing, fit-up and edge condition.	The weld symbol alone cannot create an undefined groove.
Weld symbol	System used, type, size, length, pitch, side, all-around / field flag, contour and finish.	Do not mix ISO and AWS symbol conventions without explicit instruction.
Quality requirement	Applicable welding code, quality level, WPS qualification and inspection extent.	A general “good weld” note is not an acceptance criterion.
Datums / dimensions	Pre-weld fit-up controls and final post-weld geometry.	Dimension the released state; identify machining after welding.
Distortion control	Sequence, intermittent weld, fixture or straightening requirement only when engineering-controlled.	Avoid prescribing shop sequence unless it is necessary and validated.
If the weld must be ground flush, sealed, leak-tight, dye-penetrant tested or left as-welded, say so explicitly.		

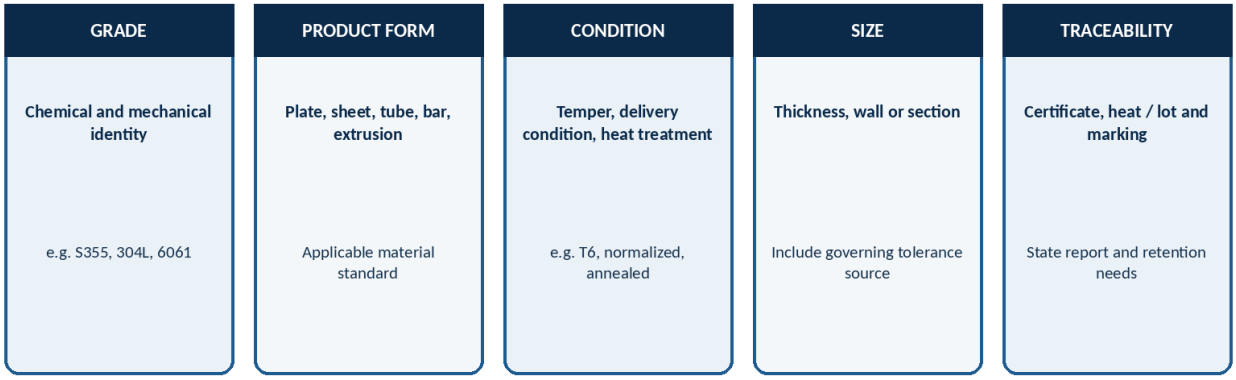
11 MATERIAL DEFINITION

Material, product form, condition and heat treatment

Material requirements must be purchasable, traceable and compatible with the manufacturing route.

Material, product form and condition

A grade name alone is rarely enough to buy the correct material.



Substitution is an engineering change when strength, formability, weldability, corrosion or finish may change.

Figure 10. A complete material callout combines grade, form, condition, size and traceability.

Requirement	Example content	Risk if omitted
Grade and standard	304L to the specified sheet / tube standard; S355 grade and delivery condition.	Nominally similar materials have different chemistry, strength and weldability.
Product form	Cold-rolled sheet, hot-finished hollow section, seamless tube, extrusion or bar.	Properties and dimensional tolerances vary by product form.
Condition / temper	Annealed, normalized, solution-treated, T6, H32 or other delivery condition.	Formability and final strength may not match design assumptions.
Heat treatment	Process, range or class, hardness / mechanical target and sequence relative to machining.	Distortion, scale, decarburization or property loss is uncontrolled.
Certification	Certificate type, heat / lot traceability, marking and retention period.	Material identity cannot be demonstrated at inspection or audit.

- Avoid commercial names unless the exact grade, governing standard and acceptable equivalents are defined.
- Any substitution that changes mechanical properties, corrosion, weldability, formability or finish response requires engineering approval.

12 SURFACE REQUIREMENTS

Surface texture, coating and cosmetic definition

Surface texture and surface treatment are different requirements and should be specified independently.

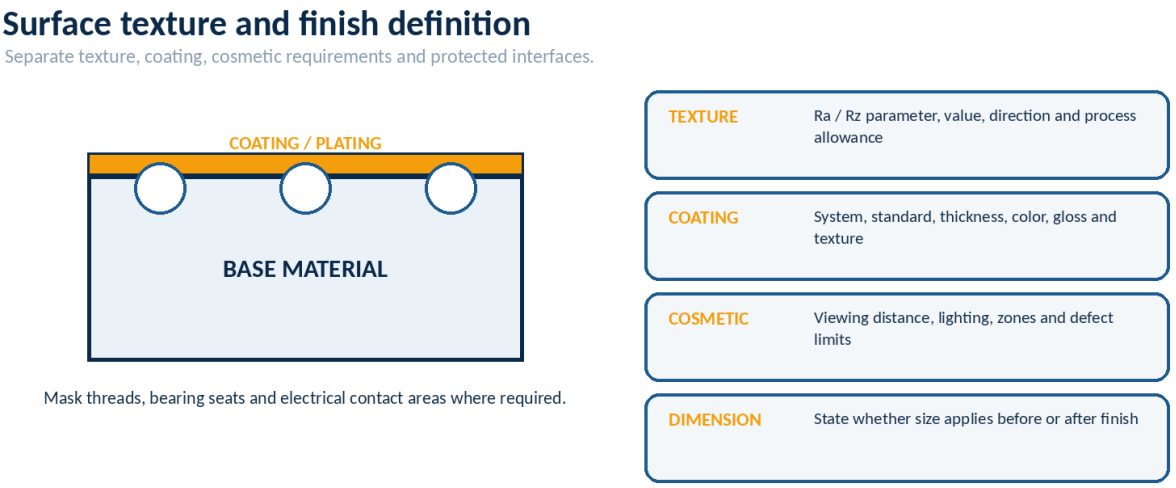


Figure 11. Define base-surface texture, coating system, cosmetic zones and protected interfaces separately.

Requirement	What to state	Verification
Surface texture	Parameter such as Ra / Rz, value, evaluation direction, cutoff / filtering basis when required, and affected surface.	Profilometer method and measurement locations.
Coating / plating	System, governing specification, class, thickness range, color code, gloss, texture and cure / seal requirement.	Thickness, color / gloss, adhesion or certificate as applicable.
Masking / no-coat	Threads, bearing seats, weld areas, electrical contacts, bonding surfaces and datum targets.	Visual inspection, thread gauge and functional fit.
Cosmetic appearance	Viewing distance, lighting, zones, defect limits, sample or approved color standard.	Controlled visual standard and zone-based inspection.
Dimensional state	Whether a size applies before finish, after finish or through a masking operation.	Final-condition dimensional inspection.

Salt-spray duration is a test condition, not a direct prediction of service life. Define the coating system and acceptance criteria, not only hours.

General tolerances, specific tolerances and GD&T

Use general tolerances for low-risk unspecified dimensions and specific controls for functional geometry.

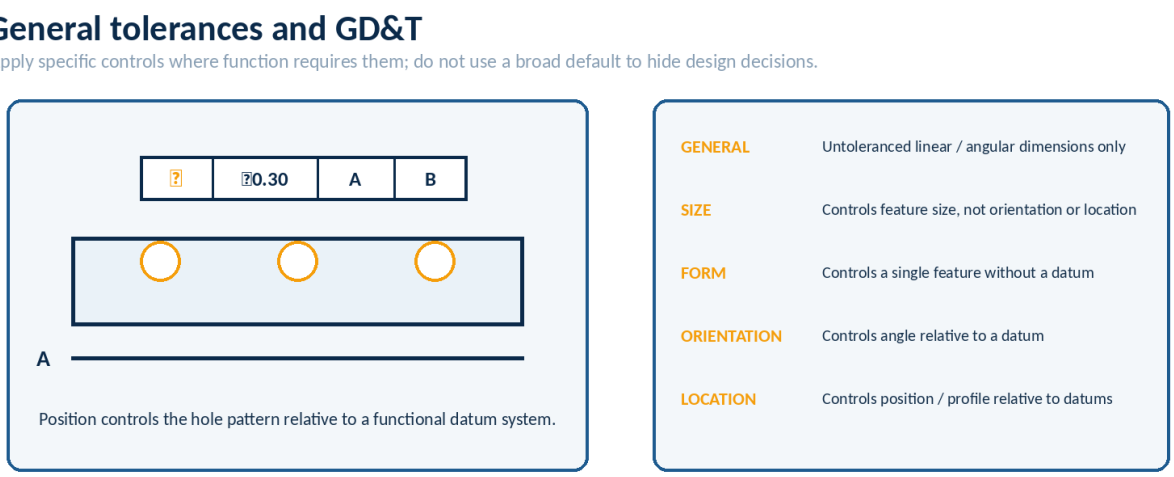


Figure 12. Size, form, orientation and location controls solve different engineering problems.

Control layer	Use	Do not assume
General linear / angular tolerance	Unspecified low-risk dimensions within the scope of the cited standard.	It applies to threads, surface texture, radii, chamfers, welded-structure geometry or explicitly controlled dimensions unless the standard says so.
Specific plus/minus tolerance	Feature size or a simple distance where the tolerance zone is clearly understood.	It controls orientation or location relative to a functional datum.
GD&T / GPS control	Form, orientation, position, runout or profile relative to a stated datum system.	Manufacturing or inspection can infer the datum sequence from the view.
Profile control	Complex contours, formed parts and weldment envelopes.	A CAD nominal model alone defines the allowed variation.
Fit / limits system	Shaft-hole, locating pin, bearing and mating-size relationships.	A clearance hole or loose fabrication feature needs a precision fit class.

Select either the ISO GPS system or ASME Y14.5 system for the drawing package and state the adopted standard and edition.

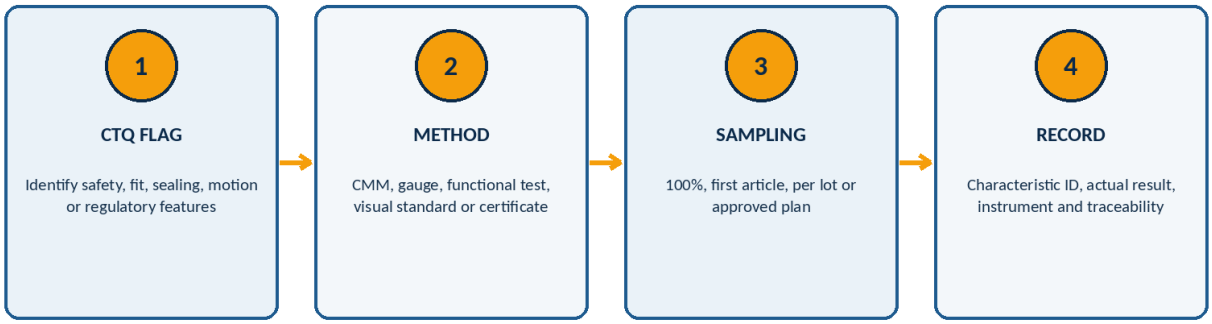
14 VERIFICATION PLANNING

Key characteristics, inspection and reporting

A requirement is not complete until there is a practical way to verify it in the released state.

Critical characteristics and inspection evidence

A drawing should make risk visible and connect each key requirement to a practical verification method.



Do not mark every dimension as critical. A key-characteristic system must remain selective and actionable.

Figure 13. Key characteristics should connect risk, method, sampling and retained evidence.

Drawing information	Examples	Control principle
Characteristic identification	Balloon / feature ID, CTQ symbol, safety or regulatory classification.	Use a controlled legend and do not mark every dimension as critical.
Inspection method	CMM, fixture gauge, thread gauge, profile template, visual standard, functional test or certificate.	Choose a method that matches the datum scheme and tolerance zone.
Sampling / frequency	First article, 100%, per setup, per lot, periodic qualification or approved sampling plan.	Critical characteristics need risk-based prevention and verification.
Report requirement	Actual values, pass/fail, instrument ID, lot, operator, date and traceability.	State when a formal dimensional, material, weld, coating or functional report is required.
Decision rule	Treatment of measurement uncertainty, guard bands or customer-specific acceptance.	Near-limit results may need more than simple rounding.

- Provide datum targets or inspection setup information when the physical datum is incomplete, flexible or inaccessible.
- Do not require a tighter measurement capability than the tolerance and risk justify.

15 CONFIGURATION CONTROL

Revision, approval and digital-file control

Every released file must represent the same approved product state and clearly show what changed.

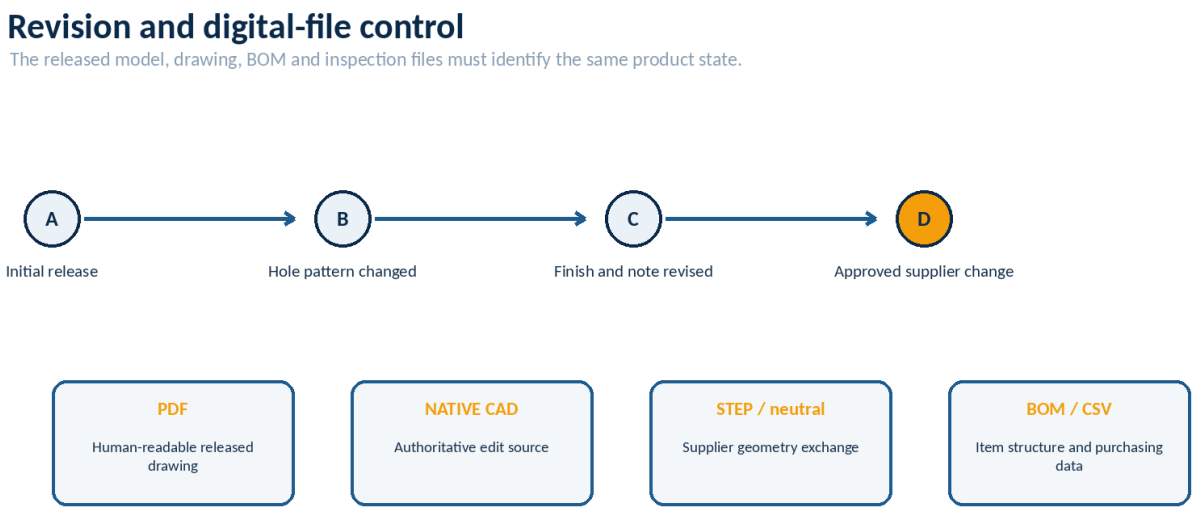


Figure 14. Revision identity must remain consistent across all drawing and digital-definition files.

Control item	Required practice	Failure mode
Part number and revision	Use a unique identity and an approved revision / lifecycle state on every released document.	Supplier quotes or builds from an obsolete file with a similar name.
Revision description	Describe the engineering change and affected requirement, not only “updated drawing.”	Reviewers cannot determine whether tooling, stock or inspection must change.
Model / drawing consistency	Regenerate views and verify dimensions, annotations and configurations after geometry changes.	The PDF, STEP and native model disagree.
File naming / transmission	Use controlled names, a release manifest and a single issue package.	Multiple email attachments become competing masters.
Neutral CAD files	State format, version, units, coordinate system and whether geometry is reference or controlling.	Scale, unit or translation errors are discovered after manufacture.
Approval / protection	Record preparer, checker, approver and release date; apply document-protection notices where required.	Unapproved edits circulate without traceability.

Superseded files should remain traceable but must not remain in the active supplier release folder.

16 RELEASE REVIEW

Manufacturing drawing release checklist

Complete the checklist against the actual PDF, model, BOM and specifications before issuing an RFQ or production release.

CHECK 01-13	CHECK 14-25
01. Part number, title and revision are unique and consistent across all files.	02. Drawing standard, units, projection method and sheet scale are stated.
03. Title block identifies material, finish, approvals and release status.	04. All required views, sections and details are present and aligned.
05. No feature is controlled by conflicting or duplicated dimensions.	06. Functional datums and the datum sequence are explicit.
07. Dimensions originate from functional interfaces or a deliberate coordinate system.	08. Reference dimensions are clearly distinguished from acceptance requirements.
09. Hole, slot, thread, counterbore and countersink callouts are complete.	10. Edge breaks, burr limits, sharp-edge and corner conditions are defined.
11. Tube stock, bend radius, angle, clocking and end features are defined.	12. Sheet-metal thickness, bends, reliefs, hardware and formed-state controls are defined.
13. Weld joint preparation, symbol system, size, length, quality and final geometry are defined.	14. Material grade, product form, condition / temper and standard are defined.
15. Heat treatment, hardness and sequence are defined where applicable.	16. Surface texture, coating, color, thickness and cosmetic criteria are separated.
17. Masking, protected threads and no-coat areas are shown.	18. General tolerance standard and edition are stated, with scope understood.
19. Specific tolerances and GD&T are matched to function and inspection.	20. Critical characteristics and report requirements are identified selectively.
21. Measurement access, datum simulation and decision rules are practical.	22. BOM balloons, item numbers, quantities and purchased-part identities agree.
23. Native CAD, neutral model, PDF and BOM carry the same revision.	24. Change description and affected tooling / inspection are reviewed.
25. DFM, quality and approval actions are closed before release.	

Release only when every “not applicable” decision has a technical reason and every open issue has an owner and due date.

17 REFERENCES AND HANDOFF

Standards reference and RFQ package

Use the edition contractually required by the customer. Verify current status before release.

Reference	Application	Official source
ISO 128-1:2020	General principles of representation	https://www.iso.org/standard/65296.html
ISO 128-2:2022	Basic conventions for lines	https://www.iso.org/standard/83355.html
ISO 128-3:2022	Views, sections and cuts	https://www.iso.org/standard/83356.html
ISO 129-1:2018	Presentation of dimensions and tolerances	https://www.iso.org/standard/64007.html
ISO 5457:1999 + Amd 1:2010	Drawing-sheet sizes and layout	https://www.iso.org/standard/29017.html
ISO 7200:2004	Title-block and document-header data fields	https://www.iso.org/standard/35446.html
ISO 1101:2017	Geometrical tolerancing fundamentals	https://www.iso.org/obp/ui/
ISO 5459:2024	Datums and datum systems	https://www.iso.org/standard/87855.html
ISO 21920-1:2021	Surface-texture indication by profile methods	https://www.iso.org/standard/72196.html
ISO 2553:2019	Welding symbols on drawings	https://www.iso.org/standard/72740.html
ISO 16792:2021	Digital product-definition data practices	https://www.iso.org/standard/73871.html
ASME Y14.100-2017	Engineering drawing practices	https://www.asme.org/codes-standards/find-codes-standards/y14-100-engineering-drawing-practices
ASME Y14.24-2020	Types and applications of engineering drawings	https://www.asme.org/codes-standards/find-codes-standards/y14-24-types-applications-engineering-drawings
ASME Y14.5-2018	Dimensioning and tolerancing	https://www.asme.org/codes-standards/find-codes-standards/y14-5-dimensioning-tolerancing

Minimum RFQ handoff

File / information	Minimum requirement
Released PDF drawing	Part / assembly number, revision, units, material, finish, tolerances and notes.
3D model when useful	Native or neutral format, units, configuration and controlling / reference status.
BOM / purchased items	Item identity, revision or approved configuration, quantity and supplier restrictions.
Process specifications	Welding, coating, heat treatment, testing and workmanship requirements.
Inspection requirements	FAI, reports, certificates, key characteristics and customer forms.
Commercial context	Quantity, annual volume, prototype / production stage, target date and packaging / logistics needs.

Do not copy proprietary standards tables into supplier documents. Cite the standard and retain a controlled licensed copy for engineering use.