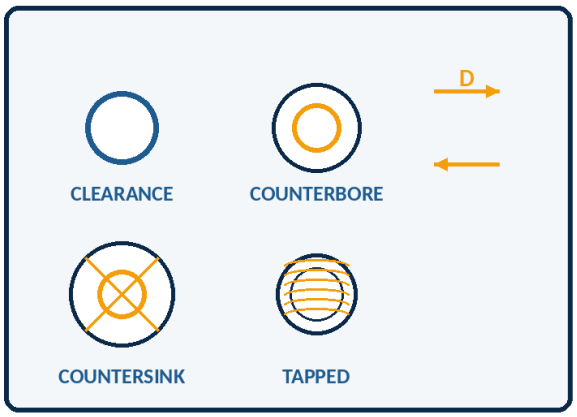
KINGSFORM METALWORKS

ENGINEERING TOOLKIT

# HOLE SIZE REFERENCE

Clearance, tapping drills, recesses, spacing, fits, location and inspection




## HOLE DEFINITION

FUNCTION — clearance, alignment, fastening

GEOMETRY — diameter, depth, angle, location

PROCESS — drill, punch, laser, ream, tap

ACCEPTANCE — size, position, finish, inspection

**A hole callout must define function, geometry, process constraints and acceptance.**

| DOCUMENT | VERSION | RELEASE DATE | LANGUAGE |
|---|---|---|---|
| KFR-012 | 1.0 | 2026-07-17 | English |

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

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

01 SCOPE AND USE

# Scope and how to use this reference

A compact engineering aid for defining holes that can be manufactured, assembled and inspected without hidden assumptions.

**Use this reference at concept review, RFQ preparation and drawing release. It is not a substitute for a controlled fastener, thread, fit or GD&T standard.**

The guide focuses on fabricated metal parts: laser-cut and punched sheet, machined plates, tubes, weldments and assemblies. It separates hole size from hole location, feature depth, recess geometry, surface condition and inspection method.

## Recommended workflow

1. Define the function: clearance, location, bearing, thread, fluid passage, plug, pin or inspection access.
2. Select the fastener, pin, insert or mating feature before choosing the hole diameter.
3. State diameter, depth, recess, angle, quantity, location, tolerance and datum references explicitly.
4. Check process capability, coating build, burr direction, access and measurement method.
5. Release one coordinated requirement across the 2D drawing, model, BOM and inspection plan.

## Contents



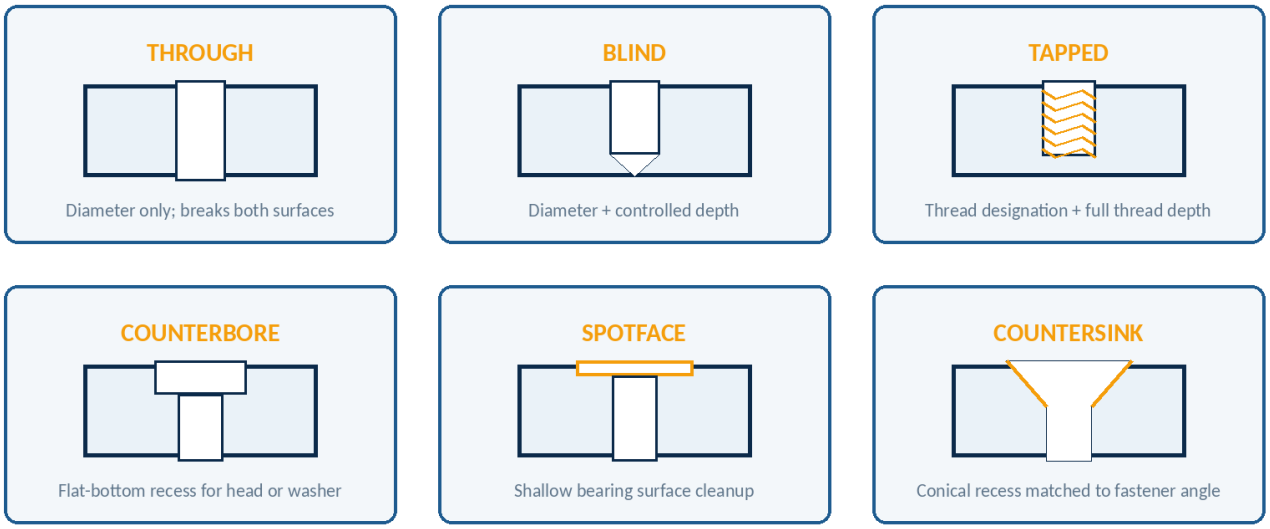
**Primary audience: designers, purchasing engineers, manufacturing engineers, quality teams and suppliers reviewing hole requirements before production.**

02 HOLE DEFINITION

# Common hole types and callout anatomy

A diameter alone rarely defines a complete manufacturing requirement.

## COMMON HOLE FEATURES




## Minimum callout content

| Feature | Minimum drawing information | Common omission |
|---|---|---|
| Through hole | Quantity, diameter or size limits, THRU, location and datum-based tolerance | No location tolerance or unclear through condition |
| Blind hole | Diameter, cylindrical depth, bottom condition if functional, location | Depth interpreted to drill point instead of full diameter |
| Tapped hole | Thread designation and class, full thread depth, drill depth, entry chamfer if required | Only thread size stated; screw bottoms before clamping |
| Counterbore / spotface | Recess diameter, depth, pilot hole, concentricity / location if functional | Head clearance or residual wall not checked |
| Countersink | Major diameter or depth, included angle, pilot hole and flushness target | Angle not matched to the fastener |

**Do not use the word “standard” without naming the standard, size series, revision or approved company specification.**

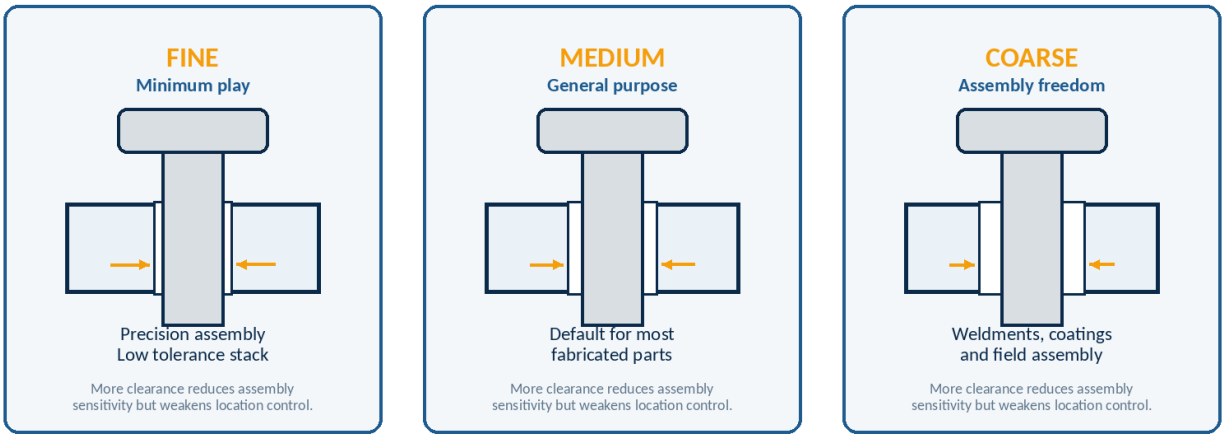
03 METRIC CLEARANCE

# Metric bolt clearance holes

Selected common ISO 273 fine, medium and coarse series values. Verify the controlled standard for contractual use.

| Fastener | Fine series mm | Medium series mm | Coarse series mm |
|---|---|---|---|
| M3 | 3.2 | 3.4 | 3.6 |
| M4 | 4.3 | 4.5 | 4.8 |
| M5 | 5.3 | 5.5 | 5.8 |
| M6 | 6.4 | 6.6 | 7.0 |
| M8 | 8.4 | 9.0 | 10.0 |
| M10 | 10.5 | 11.0 | 12.0 |
| M12 | 13.0 | 14.0 | 15.0 |
| M16 | 17.0 | 18.0 | 19.0 |
| M20 | 21.0 | 22.0 | 24.0 |
| M24 | 25.0 | 26.0 | 28.0 |

## CLEARANCE SERIES - WHAT CHANGES?




### Selection logic

- Fine: controlled alignment, low tolerance stack, minimal coating build.
- Medium: general-purpose default for machined and fabricated parts.
- Coarse: weldments, large assemblies, field installation, thick coatings or higher misalignment.

### Checks before release

- Washer bearing area remains adequate around the hole.
- Fastener head / tool access is not blocked by bends or welds.
- Tolerance stack and thermal / coating effects still permit assembly.
- Location is controlled separately from diameter.

# Inch clearance choices and functional fit selection

Use a controlled company or ASME table where contractually required; the drill choices below are practical starting points.

| Fastener | Nominal in | Close drill | Close dia | Free drill | Free dia |
|---|---|---|---|---|---|
| #4 | 0.112 | 1/8 | 0.1250 | 9/64 | 0.1406 |
| #6 | 0.138 | 9/64 | 0.1406 | 5/32 | 0.1563 |
| #8 | 0.164 | 11/64 | 0.1719 | 3/16 | 0.1875 |
| #10 | 0.190 | 13/64 | 0.2031 | 7/32 | 0.2188 |
| 1/4 | 0.250 | 17/64 | 0.2656 | 9/32 | 0.2813 |
| 5/16 | 0.3125 | 21/64 | 0.3281 | 11/32 | 0.3438 |
| 3/8 | 0.375 | 25/64 | 0.3906 | 13/32 | 0.4063 |
| 1/2 | 0.500 | 33/64 | 0.5156 | 17/32 | 0.5313 |

## Clearance hole versus fit hole

| Function | Recommended definition | Inspection approach |
|---|---|---|
| Bolt clearance | Named clearance series or explicit limits; location tolerance based on assembly stack | Pin / plug for size; CMM, vision or fixture for location |
| Dowel or locating pin | ISO 286 hole-basis fit or explicit size limits; define lead-in and surface finish | Calibrated plug / bore gauge; CMM for datum-related position |
| Press fit insert / pin | Supplier-required hole range, material hardness, wall thickness and installation method | Variable size measurement plus installation validation |
| Bearing / bushing seat | Fit class, cylindricity / position, surface texture and edge condition | Bore gauge / air gauge / CMM as appropriate |

**Coating does not “add equally” to every process. Mask or compensate threads, press-fit bores, bearing seats and precision holes using the finish supplier’s validated build data.**

05 THREADS AND TAPPING

# Tapping-drill quick reference

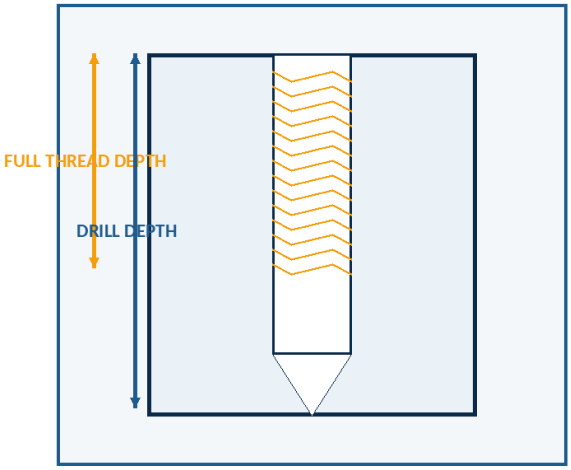
The basic cutting-tap estimate is drill diameter ≈ nominal diameter - pitch. Final selection depends on material, thread percentage, tap type and class.

| Metric thread | Typical drill mm |
|---|---|
| M2 × 0.4 | 1.6 |
| M2.5 × 0.45 | 2.05 |
| M3 × 0.5 | 2.5 |
| M4 × 0.7 | 3.3 |
| M5 × 0.8 | 4.2 |
| M6 × 1.0 | 5.0 |

| Metric thread | Typical drill mm |
|---|---|
| M8 × 1.25 | 6.8 |
| M10 × 1.5 | 8.5 |
| M12 × 1.75 | 10.2 |
| M16 × 2.0 | 14.0 |
| M20 × 2.5 | 17.5 |
| M24 × 3.0 | 21.0 |

*Typical values suit cutting taps and common materials. Form taps require different pre-hole sizes and tighter process control.*

## BLIND TAPPED HOLE - DEPTH STACK




### DESIGN STACK

1. **Required full-form thread depth**
   Functional engagement
2. **Tap chamfer allowance**
   Depends on tap style and thread size
3. **Bottom clearance**
   Prevents tap / screw bottoming
4. **Drill-point allowance**
   Adds depth below cylindrical hole

DRILL DEPTH = THREAD DEPTH + CHAMFER
+ BOTTOM CLEARANCE + DRILL-POINT ALLOWANCE
Call out thread depth and drill depth separately.

### Blind-hole callout

- Thread designation and tolerance class.
- Minimum full-form thread depth.
- Drill depth or minimum remaining wall.
- Entry chamfer, bottoming requirement and prohibited breakthrough.

### Engagement screening

- Steel threaded joint: start near 1 x nominal diameter.
- Aluminum or soft base material: often 1.5d to 2d.
- Increase only when load, fatigue, stripping or service data requires it.
- Do not rely on incomplete threads in the tap chamfer zone.

06 RECESSED HOLES

# Counterbores, spotfaces and countersinks

Design the recess around the actual fastener head, washer, tool access, coating and remaining wall thickness.

| Feature | Purpose | Starting design rule | Critical checks |
|---|---|---|---|
| Counterbore | Recess a cylindrical head or washer below / near the surface | Recess diameter = maximum head / washer diameter + assembly clearance; depth = head height + flushness allowance | Pilot concentricity, tool corner radius, remaining wall, trapped debris |
| Spotface | Create a local flat bearing surface on an uneven or coated face | Minimum diameter that fully supports the washer / nut; minimum depth needed to clean the surface | Interrupted cut, casting / weld irregularity, residual coating |
| Countersink | Seat a conical head flush or below flush | Match the included angle to the fastener; control major diameter or depth, not both unless one is reference | Knife edge, chatter, thin-sheet distortion, flushness after coating |
| Chamfer | Break an edge, guide assembly or remove burrs | State size × angle or equal-leg chamfer; identify all-around condition when intended | Do not use a generic chamfer note for functional seats |

## Fastener-angle compatibility

| Fastener family | Common included angle | Drawing note |
|---|---|---|
| Metric countersunk machine screw | 90° | Call out the applicable fastener standard or exact head geometry. |
| Unified / inch flat-head screw | 82° | Do not substitute a 90° cutter when flushness or bearing is functional. |
| Special aerospace / structural heads | 100° or other | Use the specified fastener and controlled countersink gauge / method. |

### Counterbore example callout

- 4X Ø6.6 THRU
- ⌴ Ø11.0 × 6.5 DEEP
- COUNTERBORE CONCENTRIC TO PILOT WITHIN 0.10 TIR
- DEBURR; NO RAISED EDGE ON BEARING FACE

### Countersink example callout

- 6X Ø5.5 THRU
- ⌵ Ø10.5 × 90°
- SCREW HEAD 0.0 TO 0.2 BELOW SURFACE
- VERIFY AFTER FINAL COATING

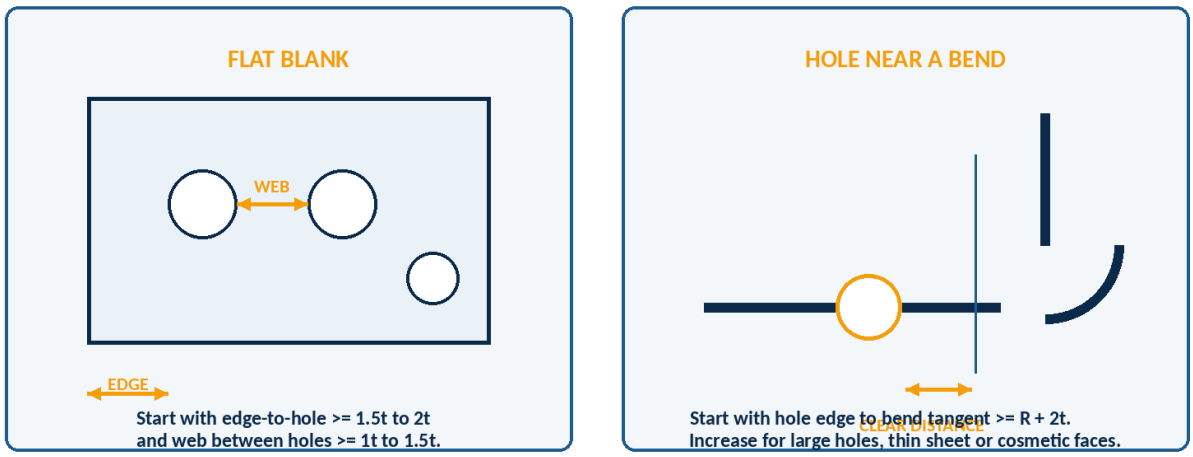
**Never dimension a functional countersink only by depth if the fastener head diameter and angle control flushness. Depth is highly sensitive to angle and tool geometry.**

07 SHEET-METAL HOLES

# Laser-cut, punched and formed-hole rules

Use these ratios as DFM screening values, then confirm with the actual material, thickness, tooling and cosmetic requirements.

## SHEET-METAL HOLE SPACING - STARTING RULES




These are DFM starting points - supplier tooling and material condition remain controlling.

| Design item | Starting point | Why it matters |
|---|---|---|
| Minimum round hole diameter | Laser: often ≥ 0.8t to 1.0t; punch: often ≥ 1.0t in mild steel and larger in harder material | Small holes increase taper, burr, tool wear and breakout risk. |
| Hole edge to part edge | Clear material from hole edge to edge ≥ 1.5t to 2t | Reduces edge distortion, breakout and weak webs. |
| Web between holes / slots | Clear web ≥ 1.0t to 1.5t | Prevents tearing, heat distortion and punch interaction. |
| Hole edge to bend tangent | Start at R + 2t; increase for large holes or cosmetic faces | Reduces ovalization, pull-in and bend-line distortion. |
| Slot end geometry | Use semicircular ends; end radius = slot width / 2 | Improves cutting and avoids internal sharp corners. |
| Formed / extruded hole | Define pilot, extrusion height, thread requirement and crack acceptance | Forming changes wall thickness and nearby geometry. |

**For coated parts, decide whether holes are cut to final size, masked, reamed after coating or accepted with coating build. State the sequence explicitly.**

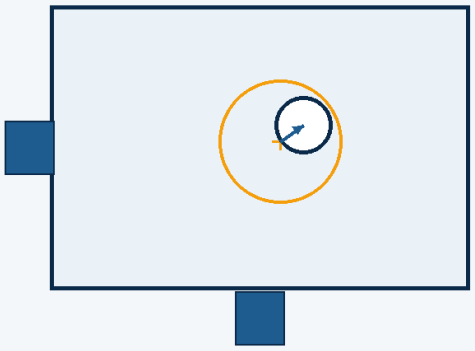
08 LOCATION AND INSPECTION

# Hole location, GD&T and measurement

A hole can meet diameter limits and still fail assembly because its axis or pattern is misplaced.

## HOLE LOCATION - SIZE IS NOT POSITION

### BASIC LOCATION + POSITION TOLERANCE



TP = 2 x sqrt(dx^2 + dy^2)

### SELECT THE METHOD BY REQUIREMENT

**PIN / PLUG GAUGE**
Fast attribute check of size or functional boundary

**CALIPER / BORE GAUGE**
Variable size measurement; access dependent

**CMM / VISION**
Location, pattern and datum-related controls

**THREAD GO / NO-GO**
Functional acceptance of internal threads

Define the measured feature, datum setup, equipment and decision rule before release.

| Requirement | Preferred control | Practical verification |
|---|---|---|
| Single clearance hole | Coordinate dimensions with explicit tolerance or position relative to functional datums | CMM, vision system or functional fixture |
| Bolt-hole pattern | Basic dimensions + position tolerance; use MMC only when functional bonus is intended | CMM / vision; functional pattern gauge where justified |
| Dowel / locating hole | Size fit + position / perpendicularity to datum system | Plug / bore gauge plus CMM or fixture |
| Tapped hole axis | Thread acceptance plus position / perpendicularity of the pitch-cylinder axis when functional | Thread gauge plus CMM, fixture or dedicated checking pin |
| Counterbore / countersink | Pilot size and location plus recess concentricity / runout or flushness requirement | Depth gauge, comparator, CMM or flushness check |

## Position calculation

- From measured coordinate errors dx and dy:
- Diametral true position = $2 \times \sqrt{(dx^2 + dy^2)}$.
- Compare the result with the allowed diametral tolerance.
- Use the stated datum reference frame and material-condition modifiers.

## Measurement planning

- Define which surface establishes each datum.
- Match gauge resolution to the tolerance and decision rule.
- Avoid caliper checks for precise internal diameters or true position.
- Record gauge ID, calibration status and actual results for critical features.

09 RELEASE CHECKLIST

# Hole-definition release checklist

A drawing is ready only when size, geometry, process effects and inspection are all closed.

| Check | Requirement | Check | Requirement |
|---|---|---|---|
| ☐ Function | Hole purpose and mating component are identified. | ☐ Size | Diameter or limit size is explicit; no ambiguous “standard hole.” |
| ☐ Quantity | Pattern quantity and equally-spaced conditions are stated. | ☐ Depth | Blind-hole cylindrical depth, drill depth and bottom condition are clear. |
| ☐ Thread | Designation, tolerance class, full thread depth and entry are defined. | ☐ Recess | Counterbore / spotface / countersink diameter, depth or angle is complete. |
| ☐ Location | Functional datums, basic / coordinate dimensions and location tolerance are defined. | ☐ Orientation | Perpendicularity or angular relationship is controlled when functional. |
| ☐ Edge / bend | Hole-to-edge, web and hole-to-bend distances have been checked. | ☐ Process | Laser, punch, drill, ream, bore, tap or form sequence is feasible. |
| ☐ Surface finish | Coating, masking, reaming-after-finish or thread protection is defined. | ☐ Burrs / edges | Burr direction and edge-break requirement protect assembly and users. |
| ☐ Access | Tooling, fastener insertion and inspection access are available. | ☐ Remaining wall | Counterbores, blind holes and intersecting features do not break through unintentionally. |
| ☐ Inspection | Method, sampling, report and gauge suitability match the requirement. | ☐ Files | 2D drawing, 3D model, BOM and inspection plan use one revision. |

## Standards and official sources

| Reference | Use | Official source |
|---|---|---|
| ISO 273:1979 | Clearance holes for bolts and screws | https://www.iso.org/standard/4183.html |
| ISO 15786:2008 | Simplified representation and dimensioning of holes | https://www.iso.org/standard/42099.html |
| ISO 129-1:2018 | Presentation of dimensions and associated tolerances | https://www.iso.org/standard/64007.html |
| ISO 286-1 / -2:2010 | ISO code system for tolerances on linear sizes and fits | https://www.iso.org/standard/45975.html |
| ISO 1101:2017 | Geometrical tolerancing | https://www.iso.org/standard/66777.html |
| ISO 5459:2024 | Datums and datum systems | https://www.iso.org/standard/87855.html |
| ISO 6410-1:1993 / -3:2021 | Representation and dimensioning of screw threads | https://www.iso.org/standard/12750.html |
| ASME Y14.5-2018 | Dimensioning and tolerancing | https://www.asme.org/codes-standards/y14-standards |

**RFQ handoff: provide the released PDF drawing, native CAD / STEP model, fastener or insert specification, finish requirement, annual quantity, inspection-report requirement and any approved deviation history.**