ENGINEERING DRAWING
Checker

FREE CHECKLIST

Mechanical drawing review checklist: 150 questions to ask before release

By Fred Vierheller, senior mechanical engineer, Certified SOLIDWORKS Professional · Updated November 3, 2026 · Versión en español

Most drawing errors aren't hard to find. They're easy to skip. This is the list we work through when we check a drawing, written so a drafter or engineer can use it on their own prints.

Each item is a question with a short answer: what to look for, why it matters, and what the fix usually is. The questions follow ASME Y14.5-2018 unless they say otherwise. A few of them, like tolerance stack-ups and motion clearance, are design questions. We've kept them on the list because someone has to own them before release, but they sit outside a drawing check.

Sections

1. Title block, document control and metadata (20)
2. Views, projection, scale and layers (20)
3. Dimensioning basics and over-dimensioning (20)
4. GD&T rules and symbols (25)
5. Datums and the datum reference frame (20)
6. Holes, threads and hardware (15)
7. Surface finish, coatings, heat treatment and edges (15)
8. Weldments, sheet metal, assemblies and parts lists (15)

Plus: the standards behind the list and how to run the check.

1. Title block, document control and metadata

Start here. If the title block is wrong, every sheet behind it is suspect.

1 ☐ **Is the governing dimensioning standard and revision year explicitly referenced in the title block or general notes?**

Look for a note like "INTERPRET DIMENSIONS AND TOLERANCES PER ASME Y14.5-2018." Without it, the shop has to guess which rules apply, and the editions really do differ: the 2018 edition dropped concentricity and symmetry, and ISO reads size and form differently than ASME. Name the standard and the year. Every time.

2 ☐ **Does the title block indicate first-angle or third-angle projection, and is the corresponding graphic symbol correct?**

Check that the projection symbol (the truncated cone) is on the sheet, and that it's drawn for the right method. North American shops expect third angle; much of Europe and Asia works in first angle. If the symbol is missing or wrong, the right-side view can get read as the left-side view, and the pocket ends up on the wrong face.

3 ☐ **Are the part name and drawing number in the title block identical to the entry in the PDM/PLM and SAP/BOM system?**

The drawing number and part name should match the PDM record and the BOM exactly, suffix and dash number included. A one-character difference is enough for purchasing to order the wrong part or for the shop to pull the wrong file. Compare them side by side; don't trust memory.

4 ☐ **Is the revision letter in the title block up to date and supported by a complete Revision History block entry?**

The revision letter in the title block should match the newest line in the revision block, and the history should run in order with no gaps. When they disagree, someone eventually machines the old geometry. Check every sheet, not just sheet 1.

5 ☐ **Does the Revision History table include the exact Document Change Notice (DCN) or Engineering Change Order (ECO) number?**

Each revision line should name the change order (ECO, ECN or DCN) that approved it. That number is how quality and the customer trace why a dimension changed. A bare "REV B, UPDATED DIMS" tells nobody anything.

6 ☐ **Is a general tolerance block defined, and are its default values appropriate for the manufacturing process?**

Make sure the tolerance block actually has values in it. A block that shows ".XX ±" with nothing after the sign means every untoleranced dimension has no tolerance at all. Then ask whether the defaults fit the process: ±.005 on three-place dimensions is easy for a CNC mill and tight for a weldment.

7 ☐ **Are linear and angular units explicitly defined on the drawing sheet?**

The sheet should say "UNLESS OTHERWISE SPECIFIED, DIMENSIONS ARE IN INCHES" (or millimeters), and angles should state degrees. Dual-dimensioned drawings need a clear rule for which unit governs. An unclear unit is a fast way to make a part 25.4 times the wrong size.

8 ☐ **Is the sheet scale noted in the title block, and does it match the actual CAD layout viewport scale?**

Check that the scale in the title block matches the views. Detail views with their own scale should carry it under the view label. Nobody should scale a print for dimensions, but a wrong scale still misleads anyone who looks at proportions.

9 ☐ **Are missing or unspecified dimensions designated as Non-To-Scale (NTS) where intentionally distorted?**

If a dimension was edited or a view was drawn out of proportion on purpose, ASME marks the value with a straight line under it rather than an "NTS" note. Either way, it has to be flagged. Better still: fix the model so the view and the number agree.

10 ☐ **Does the drawing specify material grade, specification standard, and temper/condition?**

"STEEL" isn't a material callout. You need the grade, the specification and the condition, for example "6061-T6 ALUMINUM PER ASTM B221" or "4140 STEEL, ANNEALED." The design engineer picks the material; the drawing's job is to state it completely.

11 ☐ **Are surface finish defaults specified in or adjacent to the title block for non-callout surfaces?**

A default note such as "ALL MACHINED SURFACES 125 µin Ra UNLESS OTHERWISE SPECIFIED" covers every surface that doesn't carry its own symbol. Without it, the finish on those surfaces is undefined. Check that the default makes sense for the process and that the critical faces have their own symbols.

12 ☐ **Is the signature or sign-off block complete with dates for detailer, checker, and engineering management?**

Drawn, checked and approved rows should all have names (or initials) and dates before release. Empty sign-off blocks are one of the most common findings on drawings that went out in a hurry. They also tell the customer that nobody checked the print.

13 ☐ **Does the title block contain appropriate intellectual property, proprietary data, and security classification notices?**

Check for the proprietary notice your company uses, and any export or security marking the contract requires. If the drawing carries export-controlled data, that marking matters a lot. This is a legal question your company sets; the check is only that the required notice is there and current.

14 ☐ **Are commercial or military standards called out with complete document numbers and revision levels?**

Callouts like "PAINT PER MIL-PRF-85285" or "PASSIVATE PER ASTM A967" need the full document number, and the revision or type and class where the spec has them. "Per spec" with no number is unenforceable. Confirm the spec is still active and hasn't been superseded.

15 ☐ **Is the CAD file format and native software version recorded in the document metadata?**

Some companies record the CAD system and version in the title block or the drawing properties. It helps when someone opens the file years later. It's a company rule, not an ASME one, so check it only if your drafting standard asks for it.

16 ☐ **Are weight or mass properties included in the title block, and do they reflect the actual physical model?**

If the title block has a weight field, it should come from the model's mass properties with the correct material assigned. A model left at the default material gives a wrong weight, and shipping or lifting decisions may be based on it. Blank is better than wrong.

17 ☐ **Is the sheet size correctly assigned in the page setup and boundary block?**

The sheet size letter in the title block (A, B, C, D) should match the actual sheet format. Mismatches show up when someone changes the sheet size and forgets the template. It sounds minor until a supplier prints a D-size drawing on B-size paper and can't read the notes.

18 ☐ **Are packaging, bagging, or preservation specifications referenced in the general notes?**

Parts that rust, scratch or carry precision surfaces often need a packaging or preservation note, for example "BAG AND TAG INDIVIDUALLY" or a corrosion inhibitor callout. If it matters to the customer, put it on the drawing or in the purchase order. Don't leave it to the supplier's habits.

19 ☐ **Is there a conflict between standard company drawing rules and customer-mandated drawing standards?**

When the customer has its own drafting standard, it usually wins, and it may conflict with your in-house template on title block layout, units or tolerance defaults. Decide which one governs and say so on the drawing. Mixing both is how you end up with two tolerance blocks.

20 ☐ **Are reference dimensions clearly identified with parentheses to prevent double-tolerancing?**

Reference dimensions go in parentheses, or carry "REF," and they have no tolerance. Use them for information only, like an overall length that the other dimensions already define. If a reference dimension is missing its parentheses, the shop will treat it as a toleranced dimension and you now control the same length twice.

2. Views, projection, scale and layers

Can someone who has never seen the model picture the part from these views?

21 ☐ **Do the projected views agree with the specified first-angle or third-angle projection convention?**

Lay the views next to each other and confirm they line up and read correctly for the projection you declared. In third angle, the right-side view sits to the right of the front view. If a view was moved or rotated during cleanup, mark it clearly or put it back.

22 ☐ **Does the 2D drawing geometry match the underlying 3D CAD model without manual dimension overrides?**

The drawing should be driven by the model. Watch for dimensions whose value was typed over (SolidWorks shows them in a different color), because the print then says one thing and the CAM file another. Every overridden value is a future argument with the shop.

23 ☐ **Are detail views properly labeled with an identification letter and scale ratio?**

Each detail view needs a letter on the parent view, a matching label like "DETAIL A," and the scale ("SCALE 4:1"). Check that the letters aren't reused for a section or a datum on the same drawing. Avoid letters like I, O and Q, which read as numbers.

24 ☐ **Are auxiliary views created perpendicular to angled surfaces to depict true shape and size?**

Angled features don't show true size in the standard views. An auxiliary view projected square to the angled face gives the real shape, so holes on that face can be dimensioned properly. Dimensioning a foreshortened ellipse is a common shortcut. It's also wrong.

25 ☐ **Are section views clearly cut with labeled cutting-plane lines and directional arrows?**

Section views need a cutting-plane line with arrows showing the viewing direction and letters at both ends, and a label like "SECTION B-B." Check that the arrows point the way the section actually looks. A reversed arrow flips the view.

26 ☐ **Are hidden lines shown clearly in orthographic views where internal geometry must be clarified?**

Show hidden lines when they explain something the visible lines can't, such as a blind hole or an internal slot. Often a section view does the job better. Ask: could someone tell what's inside from this view alone?

27 ☐ **Are hidden lines omitted in complex assembly or section views to prevent visual clutter?**

On busy assemblies and in section views, hidden lines usually make things worse. Leave them off unless they show something you'd otherwise miss. A view nobody can read helps nobody.

28 ☐ **Are centerlines and center marks properly placed on all cylindrical features, pitch circles, and symmetric axes?**

Every hole and cylindrical feature should have a centerline or center mark, and bolt circles should show the pitch circle. Missing center marks make it harder to see where dimensions originate. Also check for stray center marks left on features that were deleted.

29 ☐ **Is the drawing geometry placed on appropriate CAD layers rather than Layer 0 or default construction layers?**

This mainly applies to AutoCAD-style drawings. Geometry left on Layer 0 or on a construction layer may print wrong, or not at all. In SolidWorks drawings, check instead that sketch and construction geometry is hidden in the final print.

30 ☐ **Are line weights and line types formatted according to ASME Y14.2M or ISO 128 standards?**

Visible lines should be thick, hidden and center lines thin, per ASME Y14.2 (the current edition dropped the "M"). If everything prints at one weight, the drawing is hard to read on paper. Check a real print, not just the screen.

31 ☐ **Is a 3D isometric view included on the sheet to assist manufacturing in visualizing complex geometry?**

An isometric view isn't required, but it helps a lot on complex parts, especially for shops quoting quickly. Keep it small, label it as reference if you want, and don't dimension it. One picture saves a phone call.

32 ☐ **Are section hatch patterns appropriate for the specific material type?**

ASME lets you use general-purpose hatching for all materials and state the material in the title block. Material-specific patterns are optional. What matters is consistency: the same part should carry the same hatch in every section on the sheet.

33 ☐ **Are thin-walled components shown with solid filled sections rather than cross-hatching?**

Very thin sections such as gaskets, sheet metal or thin walls can be shown solid black instead of hatched. Adjacent thin parts need a small white gap between them so they still read as separate parts.

34 ☐ **Do broken-out section views clearly define boundary lines using continuous thin irregular lines?**

A broken-out section needs a thin, freehand-style break line around it, and the line should stay inside the part outline. If the boundary isn't clear, the reader can't tell where the cut stops.

35 ☐ **Are removed views placed in logical locations relative to the parent view and aligned properly?**

A removed view or removed section should sit near where it's taken from, keep its orientation where possible, and be labeled. If it has to go to another sheet, note the sheet and zone where it's taken from. Searching six sheets for "SECTION C-C" wastes the shop's time.

36 ☐ **Is drawing text scaleable and legible when printed or viewed at standard export formats like vector PDF?**

Export the PDF and print it at actual size. Text should stay sharp when zoomed in, and the smallest notes should still be readable on paper. ASME Y14.2 gives minimum lettering heights; .12 inch is common for notes.

37 ☐ **Are duplicated or overlapping geometry entities present in the CAD layout that cause double lines on export?**

Duplicate entities show up as heavy or doubled lines on export and confuse anyone measuring from a DXF. In AutoCAD, OVERKILL cleans them up. In SolidWorks, they're often leftover sketch lines visible in the drawing.

38 ☐ **Does model space geometry maintain a 1:1 scale before layout viewport scaling is applied?**

In AutoCAD workflows, the geometry should be drawn full size in model space, with scaling done only in the layout viewport. If someone scaled the geometry itself, every dimension that's measured off the model will be wrong. SolidWorks handles this for you as long as the model is right.

39 ☐ **Are phantom lines used correctly to show alternate positions, adjacent parts, or repeated features?**

Phantom lines (long dash, two short dashes) show alternate positions, adjacent parts and repeated detail. Check they're not being used where a hidden or center line belongs. Mixed-up line types make an assembly hard to read.

40 ☐ **Are break lines used properly on long uniform components to reduce drawing sheet footprint?**

Long parts with a constant section can be shortened with break lines to fit the sheet. The overall length dimension must still show the true value. Check that no feature sits inside the removed zone.

3. Dimensioning basics and over-dimensioning

Every feature needs a size and a location, given once, from the right place.

41 ☐ **Are any required functional dimensions missing, leaving a feature's location, size, or extent ambiguous?**

This is the most important check on the list, and the one software misses most. For every feature, ask: can I find its size, its location and its extent from what's on the sheet? If the shop has to measure the model or guess, a dimension is missing.

42 ☐ **Is the drawing over-dimensioned, creating closed tolerance loops that force manufacturing ambiguity?**

When the same distance can be found two ways, for example three chain dimensions plus an overall length, the tolerances fight each other. The shop can't satisfy both. Make one of them a reference dimension, or remove it.

43 ☐ **Are dimensions tied directly to functional baseline features rather than arbitrary edges?**

Dimension from the surfaces that matter to how the part works: the mounting face, the bore that locates it. Dimensions taken from a convenient edge put the tolerance in the wrong place. Datums and dimensions should tell the same story.

44 ☐ **Are dimensions placed on the view that displays the contour or true profile of the feature?**

Put the dimension where the feature shows its true shape. A hole's diameter goes on the view where it's a circle; a slot's length goes on the view where you see the slot. Dimensioning to hidden lines is a sign you're on the wrong view.

45 ☐ **Do dimension extension lines cross each other or leader lines unnecessarily?**

Extension lines can cross each other when they have to, but dimension lines shouldn't cross anything. Arrange dimensions with the shortest closest to the part and the longest outside. This one is about readability, and a messy view gets misread.

46 ☐ **Are dimensions kept outside part outlines except where legibility requires placement on the view?**

Dimensions normally sit outside the part outline. Placing one on the view is fine when it's clearer that way, for example in a crowded section. Just don't let it sit over hatching or cross a visible line.

47 ☐ **Are chain dimensions used without accounting for tolerance accumulation across the total length?**

With chain dimensions, each tolerance adds to the next, so the last feature can wander by the sum of all of them. That may be fine, but it should be a decision rather than an accident. If the end feature matters, dimension it from the start.

48 ☐ **Is baseline (datum) dimensioning applied where cumulative tolerance stackup must be minimized?**

Baseline dimensioning (every feature from one datum) keeps tolerance from piling up. Use it when features must hold their positions relative to one reference. With GD&T, the same idea becomes basic dimensions plus a position tolerance.

49 ☐ **Are basic dimensions properly enclosed in rectangular boxes to indicate theoretically exact values?**

Basic dimensions are the exact values that position or profile tolerances are measured from. They must be boxed, and they carry no tolerance of their own. A basic value without a box picks up the title block tolerance, and now the feature is controlled twice.

50 ☐ **Are duplicate dimensions applied across multiple views for the same geometric feature?**

Each dimension should appear once. If a hole diameter shows on two views, one will eventually get updated and the other won't. If you really want it visible on a second view, make it reference.

51 ☐ **Are radius callouts used for arc segments under 180 degrees and diameter callouts for complete circles?**

Arcs less than a full circle get a radius (R). Full circles and cylinders get a diameter (Ø). A full hole dimensioned as a radius is hard to measure. A fillet dimensioned as a diameter makes no sense.

52 ☐ **Is the spherical radius (SR) or spherical diameter (SØ) symbol used for spherical surfaces?**

Spherical features use SR for spherical radius or SØ for spherical diameter. A plain R on a ball end gets read as a cylinder radius. Small symbol, real difference at inspection.

53 ☐ **Are chamfer dimensions specified correctly using angle-by-length or length-by-length formats?**

A chamfer can be given as length by angle (.030 X 45°) or as two lengths. On a 45° chamfer, say whether the size is one leg or both. Chamfers at other angles need both values with tolerances, or the shop will make them 45°.

54 ☐ **Is taper or slope dimensioned using standard symbols and ratio notations per ASME Y14.5?**

Tapers use the taper symbol and a ratio (for example 0.2:1) with a size at one location. Slopes on flat surfaces use the slope symbol. A taper shown only with a few diameters at different stations usually leaves the rate of change uncontrolled.

55 ☐ **Are keyseats and keyways dimensioned with depth measured from the opposite side of the bore?**

Keyseat depth on a shaft is commonly dimensioned from the opposite side of the shaft, and keyway depth in a hub from the opposite side of the bore. That's what can actually be measured. A depth from the theoretical centerline is hard to inspect.

56 ☐ **Are undercut and relief groove dimensions fully specified with width, depth, and corner radii?**

A relief groove needs its width, depth (or diameter) and corner radii. Leaving it to "standard relief" works only if you reference which standard. Undefined grooves are often where a shaft cracks.

57 ☐ **Are angular dimensions provided with explicit tolerances rather than relying on title block linear defaults?**

Angles need tolerances too. The title block usually gives an angular default; check that it's filled in and suitable. Critical angles may need a specific tolerance, or angularity with a basic angle.

58 ☐ **Is implied 90-degree perpendicularity assumed only where lines are drawn perpendicular and governed by standards?**

Lines drawn at 90° are taken as 90° and get the general angular tolerance. That's fine for most edges. Where the right angle really matters, use perpendicularity to a datum.

59 ☐ **Are driven dimensions derived from CAD software marked as reference (REF) or enclosed in parentheses?**

Values the CAD system calculates from other dimensions, like an overall length, should be shown as reference: in parentheses or marked REF. If a driven dimension looks like a regular one, inspection will reject parts that are actually fine.

60 ☐ **Are unequal bilateral tolerances specified correctly with positive and negative deviations explicitly noted?**

Write unequal tolerances with both limits shown, for example .500 +.005/-.000, and make the number of decimals match the dimension in inch work. Watch for a zero limit that was meant to be something else. It happens more often than you'd think.

4. GD&T rules and symbols

Feature control frames are small, and small mistakes in them change what passes inspection.

61 ☐ **Does the Feature Control Frame (FCF) follow the proper order of symbol, tolerance value, modifiers, and datums?**

The order in a feature control frame is fixed: geometric symbol, tolerance value (with Ø if the zone is cylindrical), any modifiers, then datums in order of precedence. Frames built by hand in CAD sometimes get this out of order. The software also lets you leave out datums that the symbol requires.

62 ☐ **Is Maximum Material Condition (MMC) applied only to Features of Size (FOS) and never to flat surfaces?**

MMC (Ⓜ) only works on a feature of size: a hole, a pin, a slot, a tab. A flat face has no size, so Ⓜ on a flat face's flatness or perpendicularity is meaningless. If you see it, the fix is a real tolerance value instead.

63 ☐ **Is Least Material Condition (LMC) applied correctly when wall thickness or edge distance preservation is critical?**

LMC (Ⓛ) protects minimum material. Think wall thickness around a hole, or the edge distance on a boss. It's used less than MMC, so when you see it, confirm it was intended and not picked by mistake.

64 ☐ **Is the Projected Tolerance Zone symbol (P) used on threaded or press-fit holes to prevent fastener interference?**

Projected tolerance zones (Ⓟ) carry the position tolerance up above a threaded or press-fit hole, where the bolt or pin actually lives. Without it, a slightly tilted tapped hole can pass inspection and still clash with the mating part. Check that the projected height is given.

65 ☐ **Is the Continuous Feature symbol (CF) specified when interrupted surfaces function as a single continuous feature?**

CF tells the reader that interrupted surfaces, like two lands on a shaft, are to be treated as one feature. Without it, each surface gets checked on its own. Use it when function depends on them acting together.

66 ☐ **Is concentricity (◎) improperly called out on an ASME Y14.5-2018 drawing despite being removed from the standard?**

Concentricity was removed from ASME Y14.5 in 2018. If a drawing cites the 2018 edition and still uses ◎, it's citing a symbol the standard doesn't have. Position, runout or profile usually does the job, and is easier to inspect.

67 ☐ **Is symmetry (⌯) incorrectly applied on an ASME Y14.5-2018 drawing where position should be used?**

Symmetry was removed in 2018 too. Position at RFS or MMC, or profile, controls centered features. A 2018 drawing with the symmetry symbol needs to change either the symbol or the edition it cites.

68 ☐ **Is position tolerance applied to locate features of size relative to a specified datum reference frame?**

Position is the normal way to locate holes, pins and slots. The location should come from basic dimensions tied to the datums named in the frame. If the hole is also located with a ±, the drawing controls it twice.

69 ☐ **Is flat-surface profile of a surface used to control location and form without requiring a size tolerance?**

Profile of a surface with datums can control a flat face's location and form together, without a ± on the distance. The distance becomes basic. This often replaces a ± dimension plus flatness, and it's easier to read.

70 ☐ **Is profile of a line used on non-uniform or 2D contours where cross-sectional control is required?**

Profile of a line controls cross-sections one at a time, which suits extrusions and contours where each slice matters. If the whole surface matters, use profile of a surface instead. Pick based on how the part works.

71 ☐ **Is flatness applied to a planar surface without referencing datums in the feature control frame?**

Flatness controls form only, so it never takes a datum reference. A flatness frame with "A" in it is an error. If you need the face held to something else, that's parallelism or profile.

72 ☐ **Is straightness applied to a surface element vs. a feature of size axis/median line without clear distinction?**

Straightness has two forms. A frame attached to the surface controls line elements. A frame attached to the size dimension controls the axis and can take Ⓜ. Check that the placement on the drawing matches what the designer meant.

73 ☐ **Is circularity (roundness) applied to individual cross-sections of cylindrical or spherical surfaces?**

Circularity checks each circular slice on its own, with no datum. It's used on shafts and bores where roundness matters, like seals and bearings. It must be tighter than the size tolerance or it adds nothing.

74 ☐ **Is cylindricity used to control circularity, straightness, and taper simultaneously across a cylinder?**

Cylindricity bundles circularity, straightness and taper into one zone. It's harder to inspect than circularity, so use it only where the whole cylinder must be good. No datum reference.

75 ☐ **Is perpendicularity referenced to a valid datum plane or axis in the feature control frame?**

Perpendicularity needs at least one datum, and that datum must actually exist on the drawing. Check that the datum feature symbol is there. Also check the tolerance isn't zero on a flat face: a zero tolerance with no modifier can't be made or measured.

76 ☐ **Is parallelism referenced to a primary or secondary datum plane or axis?**

Parallelism also needs a datum, usually a face. It controls orientation only, not location. If the distance between the two faces matters too, a toleranced dimension or profile is still needed.

77 ☐ **Is angularity used with basic dimensions to control surface or axis orientation relative to a datum?**

Angularity needs a basic angle and a datum. A ± on the angle plus angularity is double control. Use it for angled faces or holes where orientation matters to function.

78 ☐ **Is circular runout used to control surface variations during complete 360-degree rotation?**

Circular runout checks each circular slice while the part spins on its datum axis. It catches wobble and out-of-round together. It's common on shafts and pulleys, and it's easy to check with a dial indicator.

79 ☐ **Is total runout applied across an entire surface simultaneously to control form, orientation, and coaxiality?**

Total runout applies across the whole surface at once while the part rotates. It's tighter in effect than circular runout, because it also catches taper and straightness. Use it where the full length must run true, like a bearing journal.

80 ☐ **Are tolerance zone modifiers (e.g., Free State Ⓕ, Tangent Plane Ⓣ) applied correctly where specified?**

Free state (Ⓕ) applies to flexible parts measured without restraint. Tangent plane (Ⓣ) controls only the contacting plane, not every surface point. Both are legitimate. Check that each one is used on purpose, because they loosen the requirement.

81 ☐ **Is Regardless of Feature Size (RFS) recognized as the default condition in ASME Y14.5-2009 and 2018?**

Since 1994, ASME has made RFS the default, so you don't need a symbol for it. Older drawings sometimes show Ⓢ; that's from the 1982 edition. A 2009 or 2018 drawing with Ⓢ in a position frame is a mix of editions.

82 ☐ **Are dynamic profile modifiers applied correctly to decouple form tolerance from size refinement?**

The dynamic profile modifier (a triangle, added in 2018) lets profile control form without also refining size. It's new and many shops haven't seen it. If you use it, make sure the supplier knows how to inspect it.

83 ☐ **Is a zero tolerance at MMC callout used when maximizing allowable positional tolerance based on size variation?**

Zero position at MMC is valid on features of size. The tolerance comes entirely from bonus as the hole moves away from its smallest size, so the shop gets the full size range to work with. It's a legitimate design choice, but check that the size limits are set with it in mind.

84 ☐ **Are statistical tolerancing symbols (ST) applied only when supported by formal statistical process control (SPC)?**

The statistical tolerance symbol (ST in a hexagon) only makes sense when the supplier runs statistical process control on that feature. The drawing should also carry a note explaining it. Without SPC behind it, the symbol creates confusion and nothing else.

85 ☐ **Are composite feature control frames differentiated correctly from multiple single-segment feature control frames?**

A composite frame (one symbol, two stacked rows) lets a pattern float as a group while the holes stay tight to each other. Two separate single-segment frames read differently. Make sure the drawing uses the one the designer intended, because inspection results change between them.

5. Datums and the datum reference frame

Datums tell the inspector how to hold the part. Get them wrong and good parts fail.

86 ☐ **Is the primary datum selected based on the part's primary functional mounting surface?**

The primary datum should be the surface that sets the part's orientation when it's installed, usually the largest mounting face. If the drawing makes a narrow side face primary while the part bolts down on its big face, the inspection setup won't match how the part works. Ask the designer before swapping letters.

87 ☐ **Do secondary and tertiary datums establish a fully constrained 3-2-1 spatial reference frame?**

Three mutually perpendicular datums lock all six degrees of freedom: three points on the primary, two on the secondary, one on the tertiary. Check that the frames actually reference enough datums to fix the feature. A position tolerance with only datum A lets the pattern spin.

88 ☐ **Is a datum feature symbol attached to a feature surface rather than an extension line unless controlling an axis?**

For a plane, the datum feature symbol attaches to the surface itself or to an extension line clearly offset from any dimension line. For an axis or center plane, it lines up with the size dimension. The placement is what tells the reader which one you mean.

89 ☐ **Is a datum feature symbol placed inline with a dimension line when the datum is a Feature of Size axis?**

When a hole, shaft or slot is the datum, attach the datum feature symbol in line with its size dimension or to its feature control frame. That makes the datum the axis or center plane. Attached to an edge, it would mean the surface instead.

90 ☐ **Are target datums defined on irregular or cast parts where full surface contact is impractical?**

On castings, forgings and rough parts, datum targets (points, lines or small areas) tell the inspector exactly where to touch the part. Without them, every fixture contacts different high spots and gets a different result. Each target needs a basic location.

91 ☐ **Is a cylindrical feature correctly designated as a datum axis using proper symbol placement?**

To make a cylinder a datum axis, attach the datum feature symbol to its diameter dimension or its frame. Check that the cylinder is long enough to orient the part. A thin washer face makes a poor axis.

92 ☐ **Is a pattern of holes specified as a group datum (e.g., Datum Pattern A-B) when locating secondary features?**

A hole pattern can work as one datum, for example by attaching the datum symbol to a "4X Ø.250" callout with its position frame. This matches parts located by their bolt holes. Check that the pattern itself is controlled by position before you use it as a datum.

93 ☐ **Are material condition modifiers applied to datum references in feature control frames where movable datum simulators apply?**

A modifier after a datum letter (like A-B Ⓜ) means the datum simulator can shift as the datum feature departs from its size. That adds tolerance and must match how the gauge works. Leave it out unless the part actually floats like that in assembly.

94 ☐ **Is datum precedence (Primary, Secondary, Tertiary) maintained consistently across all feature control frames?**

If one frame reads A|B|C and another reads B|A|C, the same part gets set up two different ways. Sometimes that's intended, but usually it's a mistake. Keep the order consistent unless function requires otherwise.

95 ☐ **Is the CAD model coordinate system origin mistaken for a physical datum reference frame?**

The CAD origin is wherever the modeler started sketching. It means nothing to the shop. Datums must be real features on the part, identified with datum feature symbols.

96 ☐ **Are inclined or angled surfaces used as datums without supporting angular basic dimensions?**

A datum on an angled face is legitimate, but other features need basic angles and dimensions from it. Without those, nobody can relate the angled datum to the rest of the part. Check that the angle to other datums is basic.

97 ☐ **Is a temporary manufacturing datum clearly identified for intermediate machining operations?**

Sometimes a part needs a temporary reference for a first operation, like a cast pad that gets machined off. Identify it clearly, and say whether it remains on the finished part. Otherwise inspection may look for a datum that no longer exists.

98 ☐ **Are flexible or non-rigid parts provided with a specified restraint note prior to datum inspection?**

Thin plastic, sheet metal and long tubes deflect under their own weight. A restraint note tells the inspector how to hold the part ("RESTRAIN DATUM A FLAT WITH 10 LB DISTRIBUTED LOAD"). Without it, the part passes or fails depending on how it's held.

99 ☐ **Is coplanarity established between separated surfaces using profile of a surface tied to a datum?**

Two separate faces that must be in one plane, like the mounting feet on a bracket, can be controlled with profile of a surface and a note such as "2 SURFACES" or the continuous feature symbol. That's the 2018 way to call out coplanarity. Check the faces are clearly identified.

100 ☐ **Are datum targets dimensioned with basic dimensions from existing primary datums?**

Each datum target point, line or area needs a basic location from the other datums. A target without a dimension is a guess. Target area sizes are basic too.

101 ☐ **Is a center plane datum established by referencing opposing parallel planar features?**

Two opposing parallel faces can establish a center plane, for example a slot or a tab. Attach the datum symbol in line with the width dimension. Center plane datums are common on symmetric parts, and the edition of Y14.5 matters for how you show them.

102 ☐ **Are simulated datums defined to instruct quality inspectors how to fixture the component?**

The datum feature is on the part; the datum simulator is the gauge (a surface plate, a pin, a chuck). The drawing defines the datum through its feature symbols and frames, and the inspector builds the simulator from that. If a special fixture is required, say so in a note.

103 ☐ **Is thread pitch diameter specified as the datum axis when a threaded feature serves as a datum?**

By default, a screw thread used as a datum or controlled by GD&T applies to the pitch diameter. You don't need to say "PD." Just check that this is what the designer meant.

104 ☐ **Is major or minor diameter designated as the datum axis when thread pitch diameter is not intended?**

If the major or minor diameter should govern instead of the pitch diameter, put "MAJOR DIA" or "MINOR DIA" under the frame or the datum symbol. Without that note, the pitch diameter applies.

105 ☐ **Are datum letters repeated across different features on the same drawing sheet, creating ambiguity?**

Each datum letter should be used for one feature only on the drawing. Reusing "A" for a face on sheet 2 and a bore on sheet 4 is a common mistake when parts are combined from older drawings. Also skip I, O and Q.

6. Holes, threads and hardware

Callouts have a fixed grammar. Most hole and thread findings are a missing piece of it.

106 ☐ **Does the hole callout specify drill diameter, depth, and depth symbol correctly?**

A drilled hole callout gives the diameter and, for a blind hole, the depth with the depth symbol, for example "Ø.250 ↧ .500." Through holes say THRU. Leaving off the depth on a blind hole leaves the shop guessing.

107 ☐ **Is blind hole depth measured to the full diameter shoulder rather than the drill tip point?**

Per ASME Y14.5, a hole's depth goes to the end of the full diameter, not the drill point. If the drill point matters, dimension the point angle or the point depth separately. Many first-article disputes start here.

108 ☐ **Are counterbore diameter and depth clearly called out with standard symbols and tolerances?**

Counterbores use the ⌴ symbol with diameter and depth, for example "Ø.250 THRU, ⌴ Ø.438 ↧ .250." Both values need tolerances, from the block or stated. Check that the callout sits on the view where the counterbore is visible.

109 ☐ **Are countersink diameter and included angle specified according to ASME Y14.6 standards?**

Countersinks use the ⌵ symbol with the diameter at the surface and the included angle, for example "⌵ Ø.500 X 82°." Use 82° for inch flat-head screws and 90° for metric. The symbols come from ASME Y14.5; Y14.6 covers screw threads.

110 ☐ **Does a threaded hole callout include thread series, nominal diameter, pitch/TPI, and class of fit?**

A tapped hole callout needs size, threads per inch or pitch, series, class and depth, for example "1/4-20 UNC-2B ↧ .50." A missing class is common, and it matters at inspection because the thread gauge depends on it.

111 ☐ **Is internal thread class of fit specified (e.g., 2B vs. 3B for Unified, 6H for Metric)?**

Internal threads use B in the Unified system (2B is standard, 3B is tighter) and capital letters in metric (6H is common). Leaving the class off lets the shop pick. That's usually fine until it isn't.

112 ☐ **Is external thread class of fit specified (e.g., 2A vs. 3A for Unified, 6g for Metric)?**

External threads use A (2A standard, 3A tight) or lower-case letters in metric (6g is common). Note that 2A allows a small allowance for plating; 3A doesn't. Check what the coating will do to the fit.

113 ☐ **Are thread callouts explicit about left-hand (LH) threads when non-standard right-hand threads are not used?**

Right-hand threads are the default. Left-hand threads must say LH, for example "1/2-13 UNC-2A-LH." It's rare, which is exactly why it's easy to forget.

114 ☐ **Is pipe thread geometry specified with standard designations (e.g., NPT, NPTF, BSPP, BSPT)?**

Pipe threads need the type spelled out. NPT and NPTF are tapered American threads, BSPT is tapered British, and BSPP is parallel British. They aren't interchangeable, and a wrong one leaks.

115 ☐ **Are spotface dimensions provided with diameter and depth or remaining wall thickness?**

A spotface uses the counterbore symbol with SF, plus a diameter. Give either a depth or the remaining material thickness, and say which. A spotface on a casting often reads better as "clean up" with a minimum thickness left.

116 ☐ **Is thread depth specified as minimum full thread depth rather than overall tap depth?**

For tapped holes, say how much full thread is required, for example "1/4-20 UNC-2B ↧ .50 MIN FULL THD." The tap drill goes deeper. If you only give the drill depth, the thread depth is undefined.

117 ☐ **Are hole pattern locations defined using basic dimensions tied to a common datum reference frame?**

Hole patterns should be located with basic dimensions from the datums, controlled by a position tolerance. Mixing ± locations with position on the same holes is one of the most common GD&T findings. Pick one method, and with position, that means basic.

118 ☐ **Are press-fit bore tolerances specified using ISO fit designations (e.g., H7/p6) or explicit limits?**

Press fits need the bore tolerance clearly stated: as an ISO fit like "Ø20 H7" with its limits, or as explicit limits. Check the bore and the mating pin together, because a press fit is defined by both. The interference itself is the designer's call.

119 ☐ **Is clearance hole sizing matched to standard fastener body dimensions per ASME B18.2.8?**

Clearance holes should match the fastener: ASME B18.2.8 gives normal, close and loose sizes. Pick the class on purpose and use the table value, not a rounded guess. Check the hole position tolerance still works with the clearance you picked.

120 ☐ **Are inserts specified with tap drill size, thread spec, and manufacturer part number?**

Threaded inserts (Helicoil, PEM, Keensert) need the insert's own hole size, the thread it produces and the part number. The STI tap for a Helicoil is not the same as a standard tap. Say which is meant.

7. Surface finish, coatings, heat treatment and edges

Notes that change the part after machining, and the ones people forget to write.

121 ☐ **Is surface roughness specified using Ra, Rz, or RMS values with standard units?**

Use Ra (or Rz where the customer asks for it) with units: µin on inch drawings, µm on metric. RMS is an older term you'll still see. Surface texture symbols follow ASME Y14.36. A number with no units is a guess.

122 ☐ **Does the drawing clarify whether dimensions apply before or after plating or coating?**

Plating adds thickness, so a bore that's correct before plating can be too small after. The drawing should say "DIMENSIONS APPLY AFTER PLATING" or "BEFORE PLATING." Without it, the shop and the inspector may measure at different stages.

123 ☐ **Are plating and coating specifications cited with applicable standards?**

Coating callouts need the spec and class, for example "ANODIZE PER MIL-A-8625 TYPE II CLASS 2, BLACK." "Anodize black" leaves type and thickness open. Check the spec is current.

124 ☐ **Is plating thickness specified with minimum and maximum buildup values?**

Give a thickness range, like ".0002–.0005," rather than a single number. Thin plating may not protect; thick plating can close up threads and bores. The coating spec may already set this; if so, cite the class.

125 ☐ **Are critical masking areas identified on threads, ground bores, or datum faces prior to coating?**

Threads, ground bores and datum faces often need masking before plating or paint. Show the masked areas on the drawing or in a note. If they aren't marked, assume they'll get coated.

126 ☐ **Is edge break behavior explicitly specified using 'Break all sharp edges' or ISO 13715 symbols?**

"BREAK ALL SHARP EDGES .005–.015" or an ISO 13715 edge symbol tells the shop how much to deburr. Without it, edges may be razor-sharp or overly chamfered. Some edges, like seal edges, must stay sharp; say so.

127 ☐ **Are corner fillet radii specified to prevent stress concentration in machined internal pockets?**

End mills leave a radius in inside corners. The drawing should show a radius the shop can actually cut, sized to a standard tool. A sharp inside corner either costs EDM or gets a radius the shop chooses. The radius value comes from the designer.

128 ☐ **Is material heat treatment specified with target hardness values and test standards?**

Heat treat notes need the result, like "HARDEN TO 40–45 HRC," and the test method or scale. Check whether dimensions apply before or after heat treat, because parts move. Hardness location may need to be specified too.

129 ☐ **Is case hardening depth and effective hardness specified for surface-treated steels?**

Case hardening needs both the effective case depth and the surface hardness, for example "CARBURIZE .020–.030 EFFECTIVE CASE, 58–62 HRC." Note any areas that must stay soft. These values come from the designer.

130 ☐ **Are surface texture lay directions specified where directional machining marks affect sealing or friction?**

Lay is the direction of the machining marks. It matters on seal faces and sliding surfaces, where marks running the wrong way can leak or wear. Add the lay symbol to the surface texture symbol where it matters.

131 ☐ **Is passivating, anodizing, or chemical conversion coating specified for corrosion-resistant materials?**

Stainless steel normally gets passivated (ASTM A967 or AMS 2700). Aluminum gets anodize or chromate conversion (MIL-DTL-5541). The drawing should say which, and the class. Unprotected surfaces corrode in ways nobody planned for.

132 ☐ **Are grinding relief grooves specified at shaft shoulders where mating components must seat flush?**

A shoulder with a ground diameter needs a relief groove or an undercut so the mating part (a bearing, a gear) can sit flat against the shoulder. Without it, the grinding wheel leaves a radius and the part won't seat. Check the groove is fully dimensioned.

133 ☐ **Is shot peening or stress relieving mandated in drawing notes for fatigue-critical structural parts?**

Shot peening and stress relief change fatigue life, so they belong on the drawing as a note with the spec, for example "SHOT PEEN PER AMS 2430." Whether the part needs it is the designer's decision. The check is that the note is complete.

134 ☐ **Are cosmetic surface finish standards defined for visible faces?**

If some faces are visible to the customer, say which ones and what cosmetic standard applies: no scratches over a certain size, no tool marks, a specific texture. Otherwise "cosmetic" means whatever the inspector thinks that day.

135 ☐ **Are surface finish requirements verified as realistic for the chosen manufacturing process?**

A 16 µin Ra finish on a sand casting won't happen without extra operations. Check that each finish requirement fits the process that makes the surface. Where it doesn't, it needs to be a conscious, costed decision.

8. Weldments, sheet metal, assemblies and parts lists

The last stretch: weld symbols, sheet metal, and making the parts list match the picture.

136 ☐ **Are weld symbols specified according to AWS A2.4 or ISO 2553 standards?**

Weld symbols follow AWS A2.4 in the US or ISO 2553 elsewhere. The two differ in how they show arrow side and other side. Make sure the drawing names the standard, and that the symbols are drawn to it.

137 ☐ **Do weld callouts specify weld type, size, length, pitch, and fill condition?**

A weld symbol needs the weld type (fillet, groove), size, length and, for intermittent welds, pitch. "Weld all around" needs the circle. Missing sizes are the most common weld finding.

138 ☐ **Are non-destructive examination (NDE) requirements specified for structural welds?**

If welds must be inspected beyond visual, say how: dye penetrant, magnetic particle, radiography, ultrasonic. Add the acceptance standard (for example AWS D1.1). The level of inspection is a design decision; the drawing must state it.

139 ☐ **Is sheet metal bend radius specified as inside bend radius (IBR) consistent with tooling standards?**

Bend radius is usually given as the inside radius, and it should match tooling the shop has. Tight radii crack hard tempers like 6061-T6. Check the radius is stated, not assumed.

140 ☐ **Are sheet metal bend relief slots dimensioned at corners to prevent material tearing?**

Where a bend meets an edge, a small relief cut keeps the metal from tearing. Check the relief is shown and dimensioned on the flat pattern or the formed view. CAD adds it automatically; make sure it survives.

141 ☐ **Is grain direction specified for sheet metal or plate components subject to high bending stresses?**

Steel plate and aluminum sheet bend more easily across the grain than with it. If a part has tight bends, the drawing may need a grain direction arrow. That's a design decision; the check is that it's shown when needed.

142 ☐ **Are flat pattern dimensions provided as reference only to allow supplier K-factor adjustment?**

Flat pattern dimensions should be marked reference, because each shop uses its own bend allowance or K-factor. The formed part is what gets inspected. A toleranced flat pattern forces the shop to match your CAD's K-factor.

143 ☐ **Does the assembly Bill of Materials (BOM) list item numbers, part numbers, descriptions, quantities, and materials?**

A complete parts list has item number, part number, description and quantity, and usually material. Check that descriptions are real names, not CAD file names. A parts list that reads "Part1^Assem3" isn't a parts list.

144 ☐ **Do item find numbers (balloons) on the assembly view match the BOM table line items exactly?**

Every item number in the parts list should have a balloon, and every balloon a parts list entry. Check the numbers match exactly. Count them; it's quick.

145 ☐ **Are standard off-the-shelf hardware items specified with complete vendor COTS part numbers?**

Purchased parts (screws, pins, bearings) need a vendor part number or standard, for example "SHCS 1/4-20 X .75 LG, 91251A540." "Bolt" is not enough. Check the length and grade too.

146 ☐ **Is torque or assembly fastener installation specification referenced in assembly notes?**

If fasteners need a specific torque, put it on the assembly drawing or reference the assembly spec. Leaving it out means torque by feel. Torque values come from the designer.

147 ☐ **Are lubrication, adhesive, or threadlocker application requirements clearly called out?**

Threadlocker, grease or adhesive should be called out by product or spec (for example "LOCTITE 242") and where it goes. Assembly techs will otherwise use whatever's on the bench.

148 ☐ **Are dynamic sub-assemblies checked for physical interference and clearance throughout full motion envelopes?**

This is a design check, not a drawing check: it happens in CAD motion studies or prototypes. A drawing review won't find it. Make sure someone owns it before release.

149 ☐ **Is a concession, deviation, or engineering variance form required when drawings deviate from production standards?**

If a part has to deviate from the drawing or a company standard, the deviation should be approved on paper and tied to the part. Don't fix it by editing the drawing quietly. The deviation process protects everybody.

150 ☐ **Has an independent engineering check been conducted by a qualified reviewer other than the original CAD detailer?**

Yes, and this one makes the other 149 stick. The drafter knows what they meant, so they read their own drawing that way. An independent checker reads only what's on the sheet, which is exactly what the shop will do.

The standards behind the list

The documents most of these questions come back to. Your company or your customer may add their own drafting manual on top.

Standard	Covers	Worth knowing
ASME Y14.5	Dimensioning and tolerancing, GD&T	The 2018 edition removed concentricity and symmetry. Cite the edition.
ASME Y14.100	Engineering drawing practices	Sheet formats, revisions, parts lists, drawing types.
ASME Y14.2	Line conventions and lettering	Line types, line weights, text height.
ASME Y14.36	Surface texture symbols	How finish symbols are drawn. ASME B46.1 defines the parameters (Ra, Rz).
ASME B18.2.8	Clearance holes for fasteners	Close, normal and loose hole sizes.
ISO 1101 / ISO 2768	ISO geometric tolerancing / general tolerances	ISO reads size and form differently from ASME. 2768 classes: f, m, c, v.
ISO 13715	Edges of undefined shape	Edge break and burr symbols.
AWS A2.4	Welding symbols	US weld symbol rules. ISO 2553 is the international version.

How to run the check

Don't read 150 questions top to bottom against a whole drawing set. Work in passes, one topic at a time, across every sheet. You'll catch more, because your eye is tuned to one kind of error.

1. **Title block and records.** Standard cited, units, revision, part number against PDM and BOM, sign-offs.
2. **Views.** Projection, section and detail labels, hidden lines, what prints on the PDF.
3. **Dimensions.** Missing sizes and locations, double dimensions, reference values, chains.
4. **GD&T and datums.** Frame grammar, datums identified and complete, basic dimensions, modifiers.
5. **Holes, threads and finishes.** Callouts, thread class, coatings, heat treat, edge breaks.
6. **Assembly and sign-off.** Balloons against the parts list, purchased parts, notes, and an independent check.

And have someone other than the drafter do it. The person who drew it reads what they meant. A checker reads what's there, which is what the shop will do. More on setting this up: how to check a mechanical drawing.

Want a second set of eyes on your drawings?

Fred checks your PDF drawings against ASME Y14.5 and sends a findings report: every correction located by sheet and zone, with what to change and why. From $29 a sheet. NDA first.

engineeringdrawingchecker.com · (817) 736-1783 · hola@engineeringdrawingchecker.com