SAND2021-15161PE

Sandia National Laboratories

Exceptional service in the national interest

Intro to Drawings and Geometric Dimensioning & Tolerancing (GD&T)

Per ASME Y14.5-2009

Caleb White

12/02/2021

Sandia National Laboratories is a multimission laboratory managed and operated by National Technology and Engineering Solutions of Sandia LLC, a wholly owned subsidiary of Honeywell International Inc. for the U.S. Department of Energy's National Nuclear Security Administration under contract DE-NA0003525.

Sandia National Laboratories is a multimission laboratory managed and operated by National Technology & Engineering Solutions of Sandia, LLC, a wholly owned subsidiary of Honeywell International Inc., for the U.S. Department of Energy's National Nuclear Security Administration under contract DE-NA0003525.

About Me

BS & MS in Mechanical Engineering

DPM on LMS19

Undergrad Intern (2017)

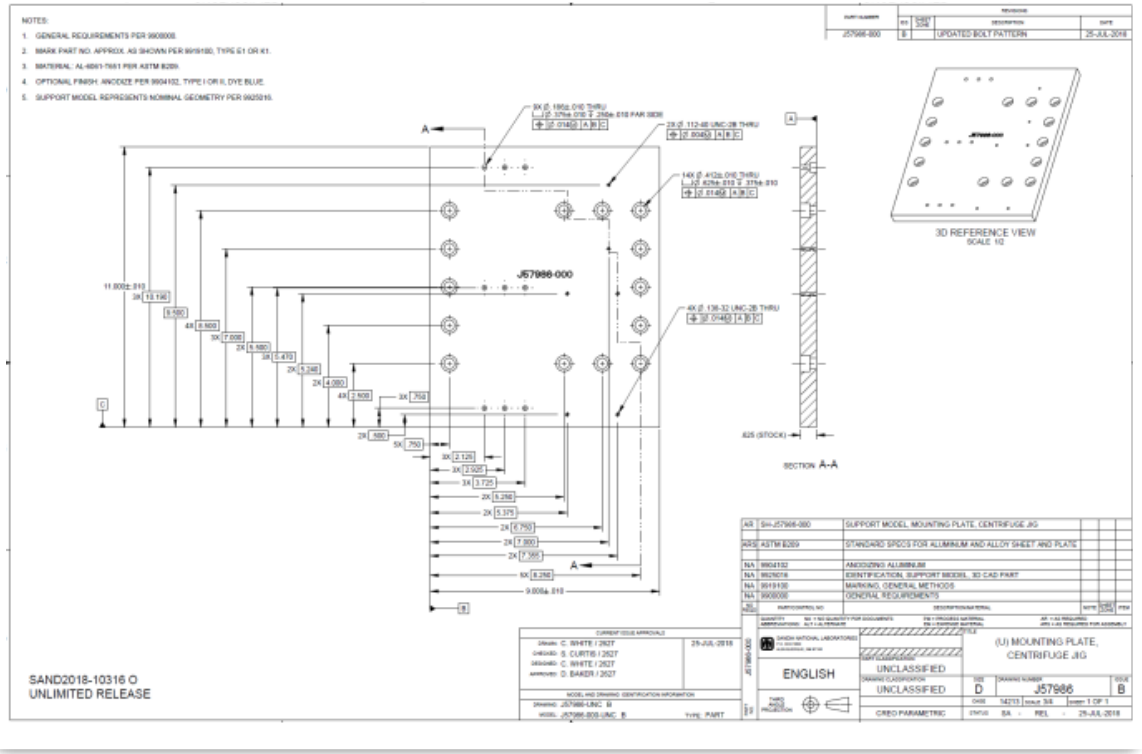
Virtual Worker in Tyler, TX

Y14.5-2009 Technologist Certification

The Secretary's Achievement Award (2021)

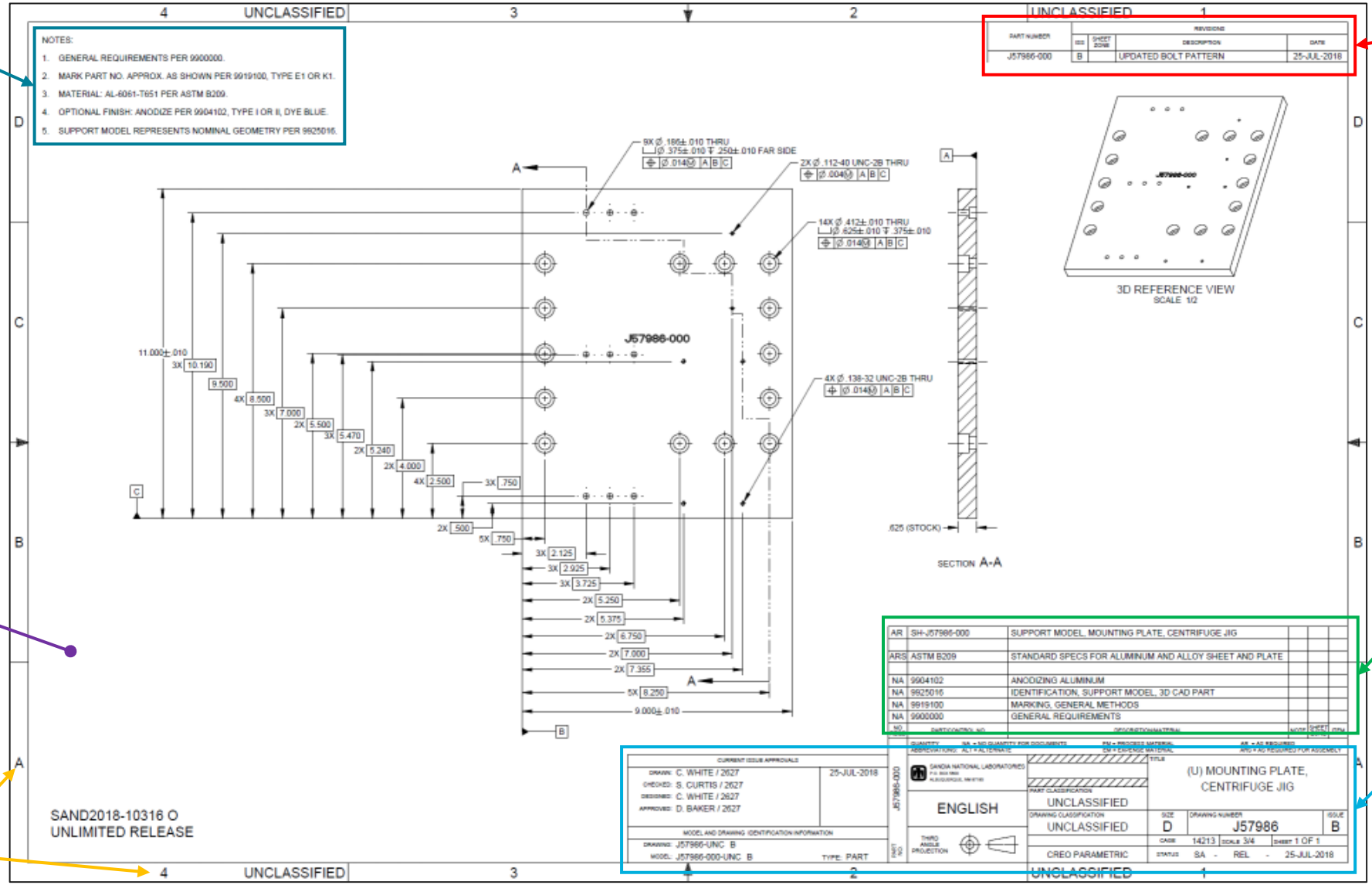
Importance of Engineering Drawings

- Technical requirements document.
  - Fully defines all details critical to the *fit, form, or function* of the design/product.
  - Specifies: geometry, dimensions, tolerances, materials, finishes, specifications, Bill of Materials (BOM), instructions, additional processes, etc.
- Legal instrument in contractual agreement between designer and manufacturer.
  - Manufacturers are free from liability for faulty drawings, designers have legal recourse for manufacturing mistakes.



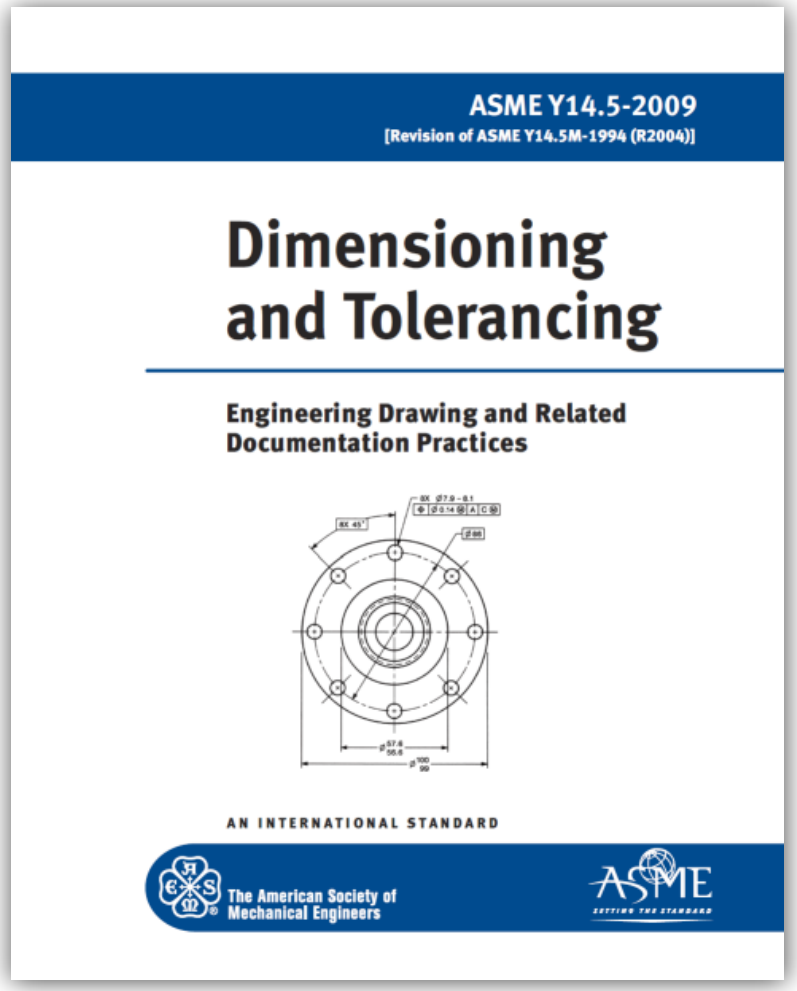
Incomplete/inaccurate drawings can cause severe production delays and increased costs.

Drawing Components



Geometric Tolerancing & Dimensioning (GD&T)

- Credited to Stanley Parker in 1940.
- International standard that uses symbolic language on models and drawings to define nominal (theoretically perfect) geometry and the allowable variation in
  1) form and possible size of individual features,
  2) between features.
- Defines references coordinate system(s) used throughout manufacturing and inspection; concisely describes complex requirements.
- Greatly simplifies tolerance analyses.
- Assures interchangeability.
- Related standards:
  - ASME Y14.5.1M *Mathematical Definition of Dimensioning and Tolerancing Principles*
  - ASME Y14.41 *Digital Product Definition Data Practices*



GD&T defines the theoretically exact part and the maximum amount of allowable variation—great emphasis on *design intent*.

Fundamental Rules

- All dimensions *shall* have a tolerance.
  - Applied directly, indirectly (basic), general note, or tolerance block.
  - Exceptions for reference, max, min, and stock.
- Dimensioning/tolerancing *shall* be complete for full understanding of every feature. Measurement and scaling of the drawing is disallowed*.
- All necessary dims *shall* be shown (no more than needed for completeness), reference dims should be minimized.
- Dims should be arranged to suit function/mating of part. Dims *shall* not be subject to more than one interpretation.
- Manufacturing methods should not be specified unless essential to requirements
- Non-mandatory processing dims are marked as such (NONMANDATORY), if final dims are given.
- Dims should be arranged for optimum readability and be applied in true profile views that refer to visible outlines.
- Geometry (i.e., wires, etc.) manufactured to gage or code numbers *shall* be specified by linear dimensions with the Gage or code numbers shown in following parentheses.
- ∠(0°, 90°) basic angles are assumed for lines with no angular dim, tolerance is provided by feature control frames governing the feature.
- All dims/tolerances apply at 20°C (68°F) and 101.3 kPa (14.69 psi), unless stated.
- Unless specified , all tolerances apply to full depth, width, and length of feature.
- Dims only apply at specified drawing level.
- If coordinate system is shown: it *shall* be right-handed, show axis labels, and positive direction.

Standard must be invoked on drawing: DIMINSIONING AND TOLERANCING PER ASME Y14.5-2009.

Cannot mix standards (i.e., 1994 and 2009) on same drawing.

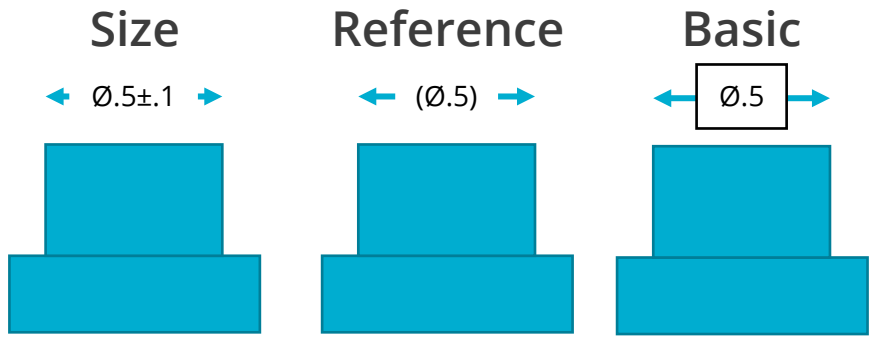
Dimensions & Tolerances

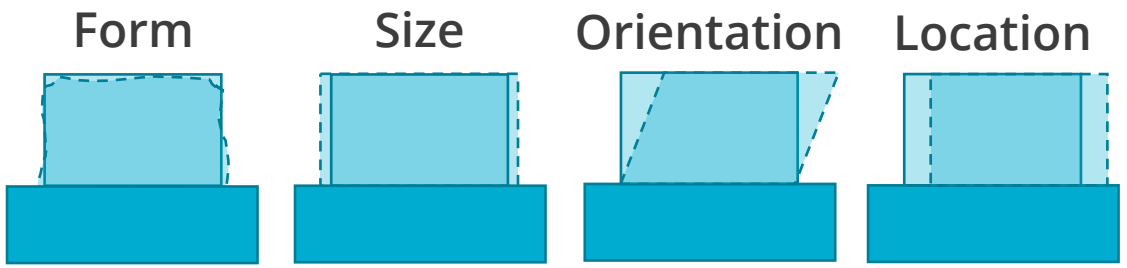
Dimension: a numerical value(s) or mathematical expression in appropriate units of measure used to define the *form, size, orientation, or location*, of a part or feature.

Tolerance: the total amount a specific dimension is permitted to vary. The tolerance is the difference between the maximum and minimum limits.

Feature: a physical portion of a part such as a surface, pin, hole, or slot or its representation on drawings, models, or digital data files.

Regular Feature of Size: one cylindrical or spherical surface, a circular element, and a set of two opposed parallel elements or opposed parallel surfaces, each of which is associated with a directly toleranced dimension.



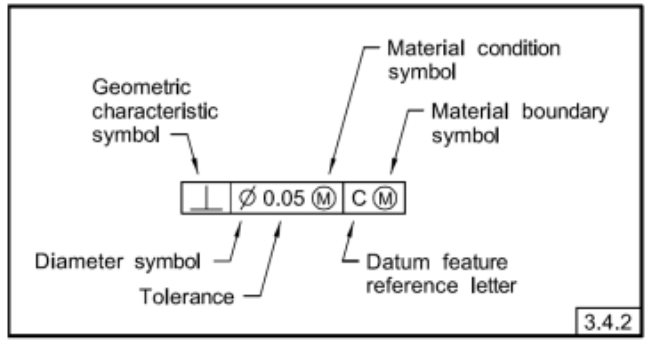


Directly Toleranced

Ø.5±.1 Ø.6 Ø.4 Ø.5 $^{0}_{-.1}$

UNLESS OTHERWISE SPECIFIED:
LINEAR TOLERANCE: ±.2
ANGULAR TOLERANCE: ±1°

Indirectly Toleranced



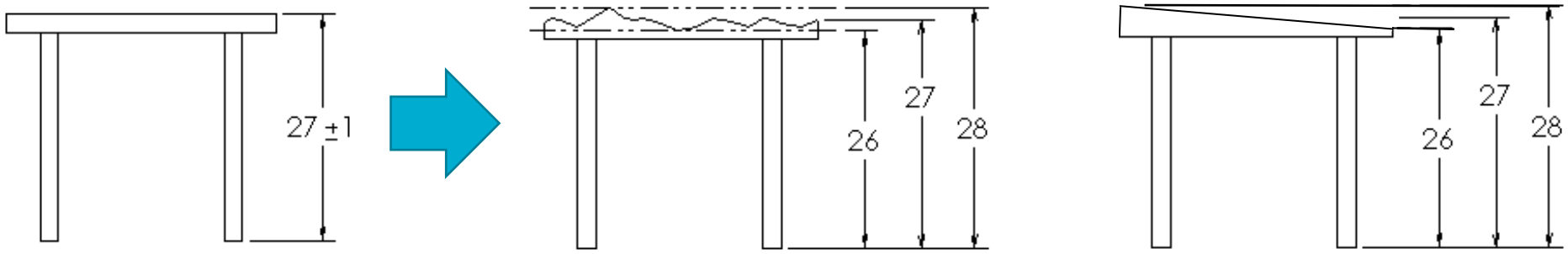
Rule #1: The form of an individual regular feature of size is controlled by its limits of size (perfect form at MMC).

Table Design Example

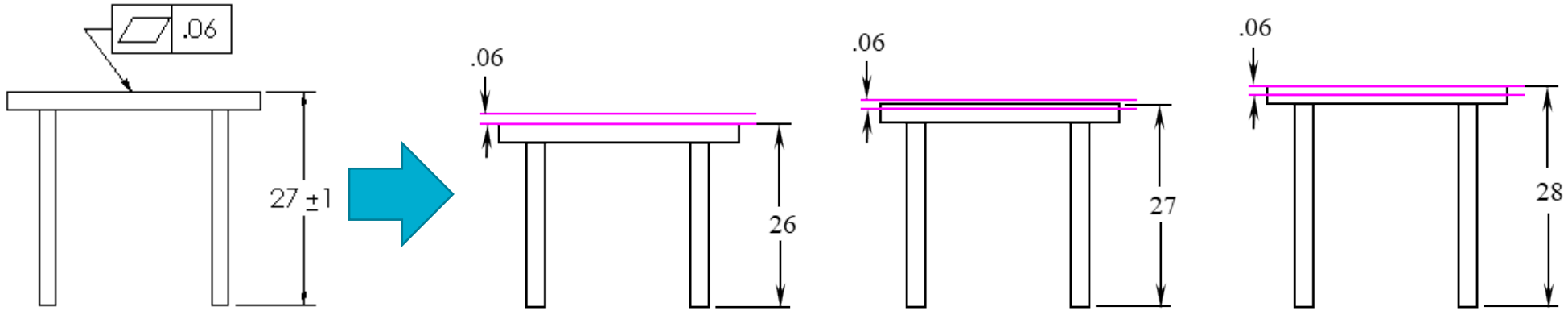
Requirements

Design a table that stands between 26–28” high and is flat to within 1/16” (≈.06”).

Direct Tolerancing
Per Rule #1, Form is given from Size tolerance.



Indirect with GD&T
Form and Size can be controlled separately.



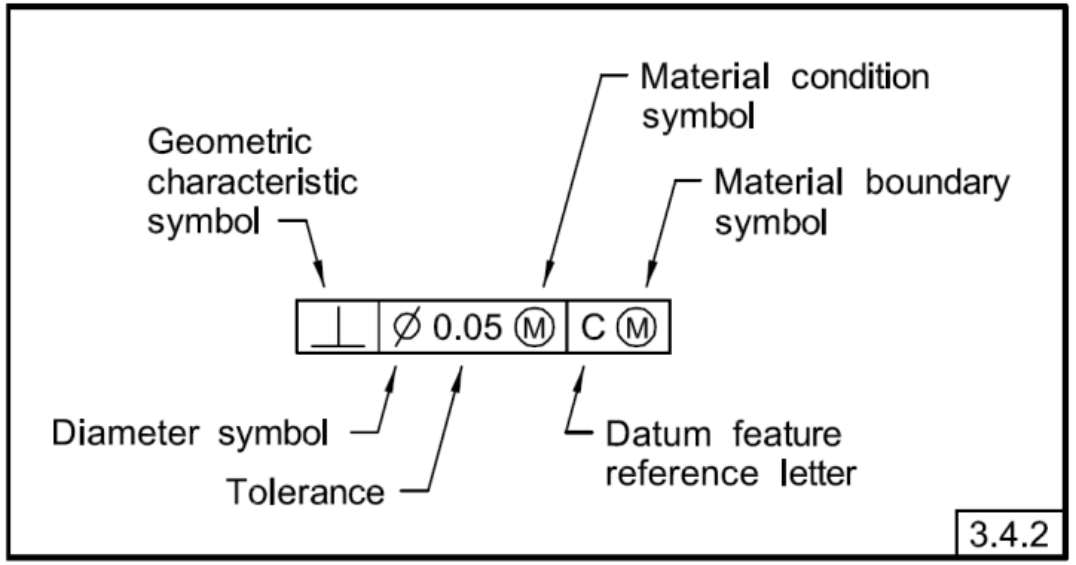
Geometric Symbols

APPLICATION	TYPE OF TOLERANCE	CHARACTERISTIC	SYMBOL	SEE:
INDIVIDUAL FEATURES	FORM	STRAIGHTNESS		5.4.1
		FLATNESS		5.4.2
		CIRCULARITY		5.4.3
		CYLINDRICITY		5.4.4
INDIVIDUAL OR RELATED FEATURES	PROFILE	PROFILE OF A LINE		8.2.1.2
		PROFILE OF A SURFACE		8.2.1.1
RELATED FEATURES	ORIENTATION	ANGULARITY		6.3.1
		PERPENDICULARITY		6.3.3
		PARALLELISM		6.3.2
	LOCATION	POSITION **		7.2
		~~CONCENTRICITY~~		7.6.4
		~~SYMMETRY~~		7.7.2
	RUNOUT	CIRCULAR RUNOUT		9.4.1
		TOTAL RUNOUT		9.4.2

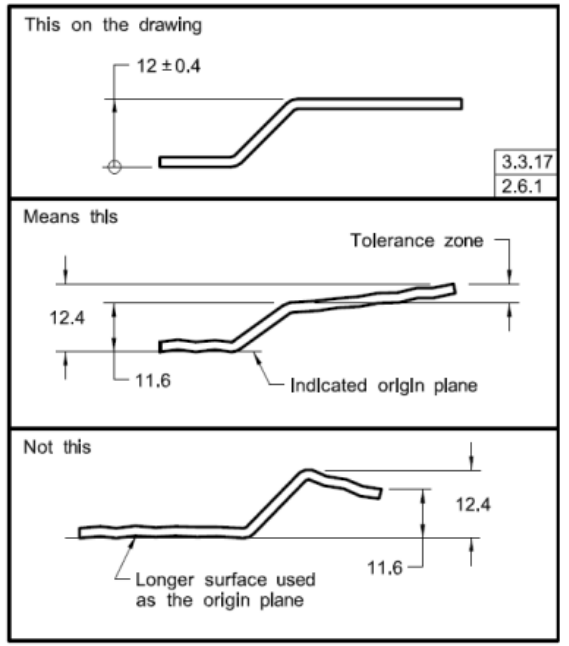
* Arrowheads may be filled or not filled ** May be related or unrelated 3.3.1

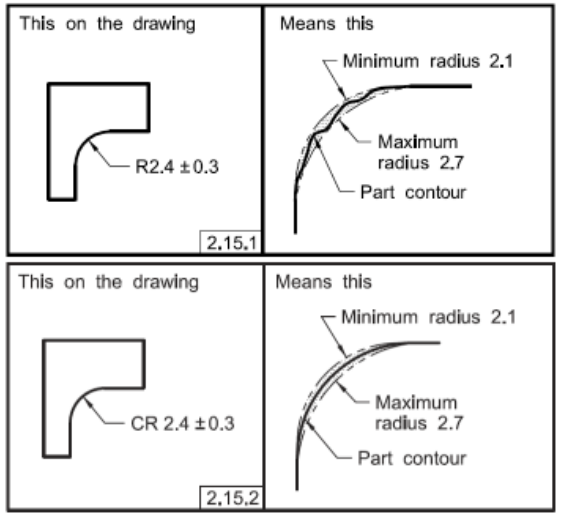
Modifying Symbols

TERM	SYMBOL	SEE:
AT MAXIMUM MATERIAL CONDITION (When applied to a tolerance value) AT MAXIMUM MATERIAL BOUNDARY (When applied to a datum reference)	Ⓜ	3.3.5
AT LEAST MATERIAL CONDITION (When applied to a tolerance value) AT LEAST MATERIAL BOUNDARY (When applied to a datum reference)	Ⓛ	3.3.5
TRANSLATION	▷	3.3.26
PROJECTED TOLERANCE ZONE	Ⓟ	3.3.6
FREE STATE	Ⓕ	3.3.20
TANGENT PLANE	Ⓣ	3.3.21
UNEQUALLY DISPOSED PROFILE	Ⓤ	3.3.22
INDEPENDENCY	Ⓘ	3.3.24
STATISTICAL TOLERANCE	ST	3.3.10
CONTINUOUS FEATURE	CF	3.3.23
DIAMETER	∅	3.3.7
SPHERICAL DIAMETER	S∅	3.3.7
RADIUS	R	3.3.7
SPHERICAL RADIUS	SR	3.3.7



CONTROLLED RADIUS	CR	3.3.7
SQUARE	□	3.3.16
REFERENCE	( )	3.3.8
ARC LENGTH	⌒	3.3.9
DIMENSION ORIGIN	⊕→	3.3.17
BETWEEN	←→	3.3.11
ALL AROUND		3.3.19
ALL OVER		3.3.25



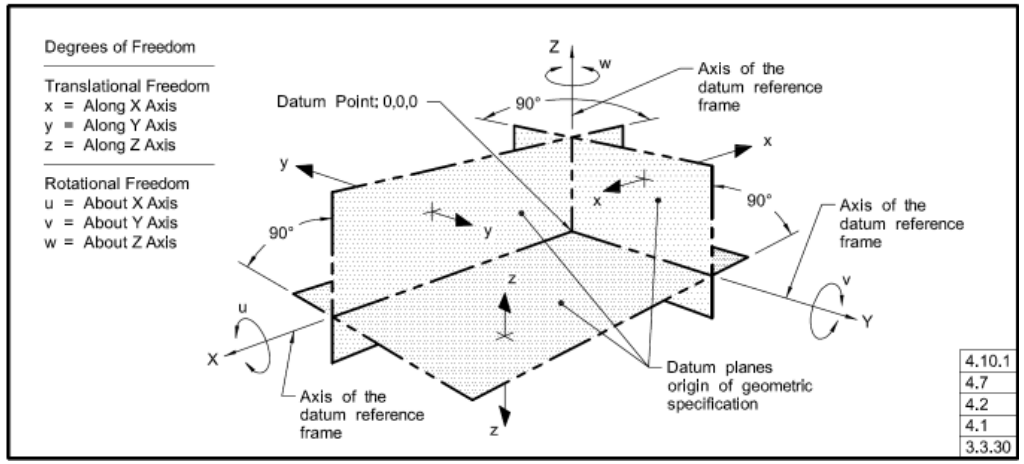


Datums & Datum Features

Datum: a theoretically exact point, axis, line, plane, or combination thereof derived from the theoretical datum feature simulator.

Datum Feature: a feature that is identified with either a datum feature symbol or a datum target symbol.

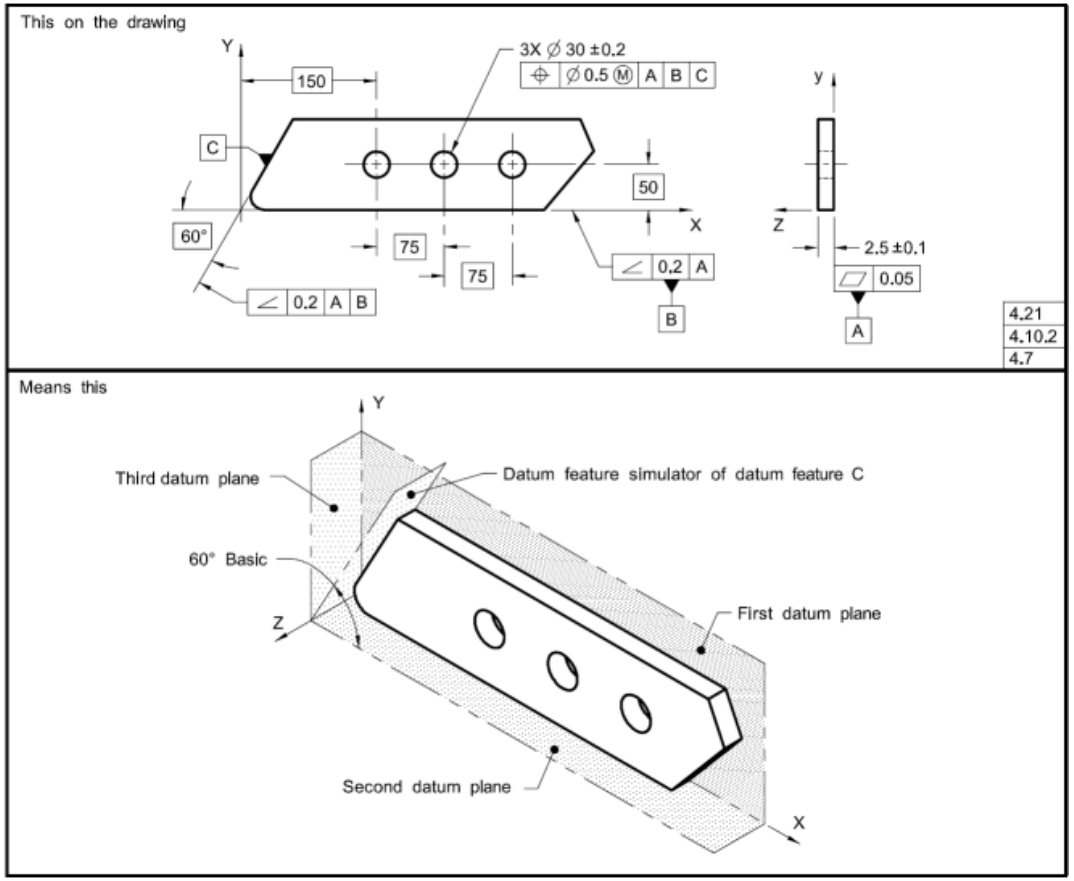
Datum Feature Simulator: the physical boundary used to establish a simulated datum from a specified datum feature.

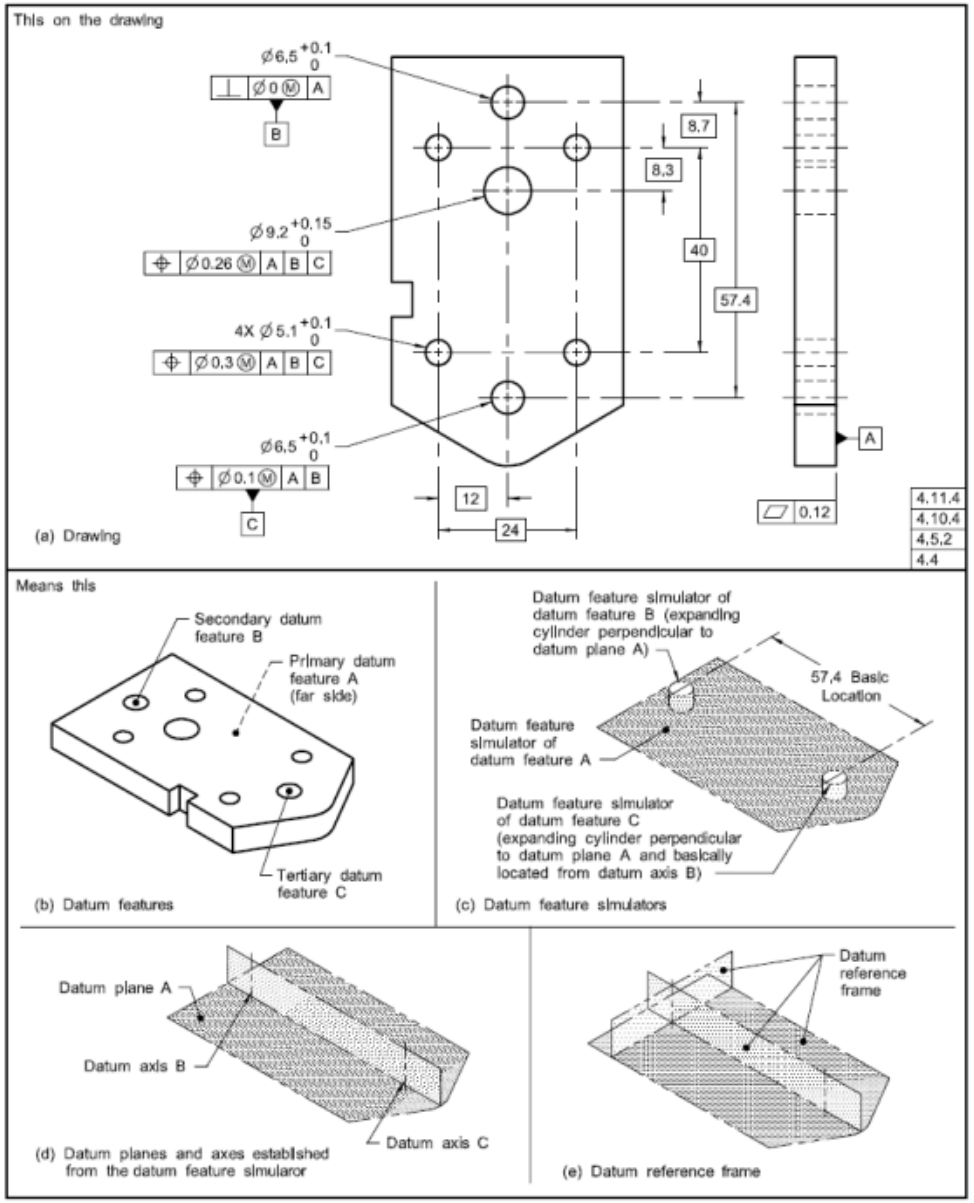


FEATURE TYPE	ON THE DRAWING	DATUM FEATURE	DATUM AND DATUM FEATURE SIMULATOR	DATUM AND CONSTRAINING DEGREES OF FREEDOM
PLANAR (a)			PLANE	
WIDTH (b)			CENTER PLANE	
SPHERICAL (c)			POINT	
CYLINDRICAL (d)			AXIS	
CONICAL (e)			AXIS & POINT	
LINEAR EXTRUDED SHAPE (f)			AXIS & CENTER PLANE	
COMPLEX (g)			AXIS, POINT, & CENTER PLANE	4.23 4.3 4.11.4 4.2

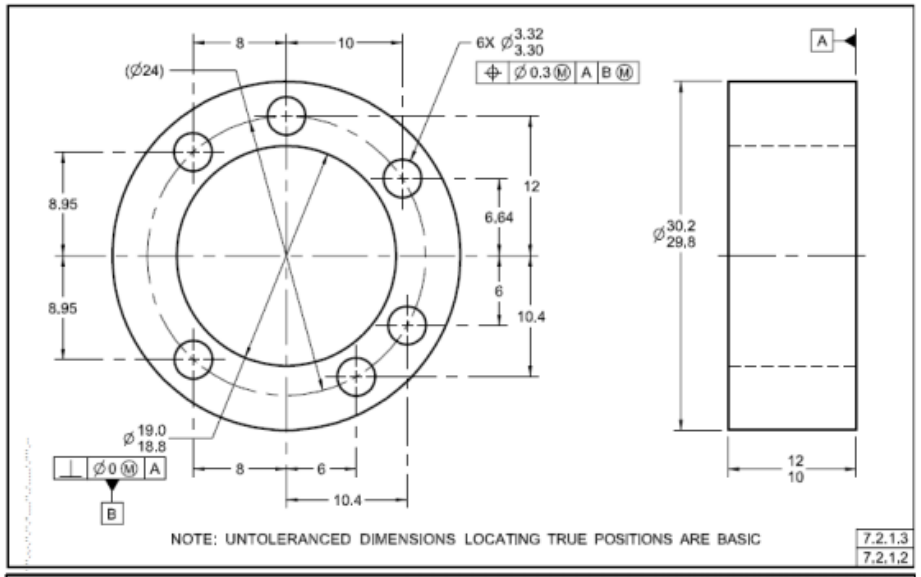
Developing Datum Reference Frame

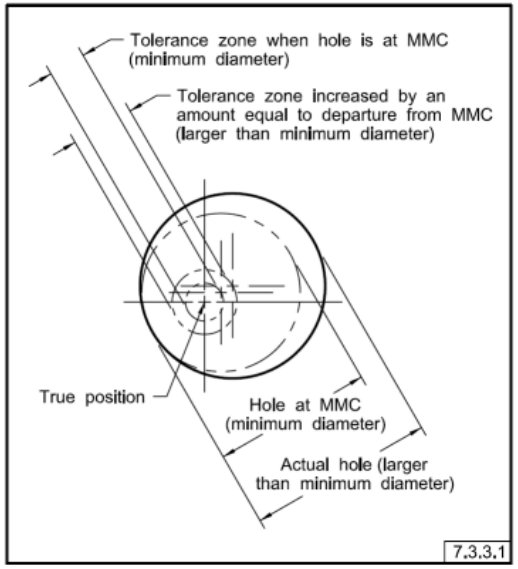
1. Identify datum features.
2. Relate (control) datum features to each other.
3. Locate surfaces and features of size.
4. Refine orientation and form if necessary.

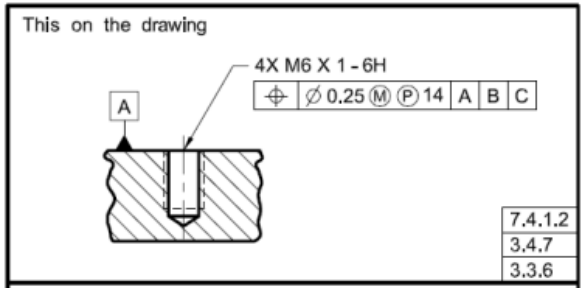


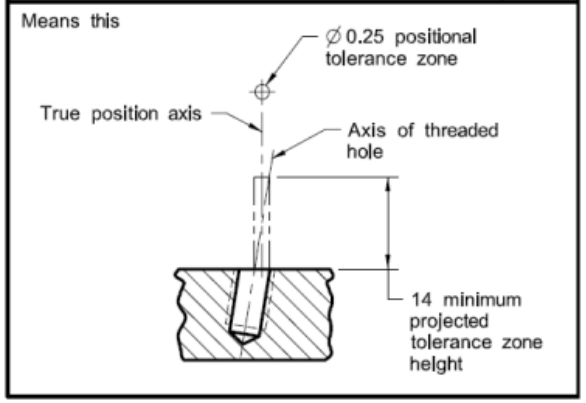


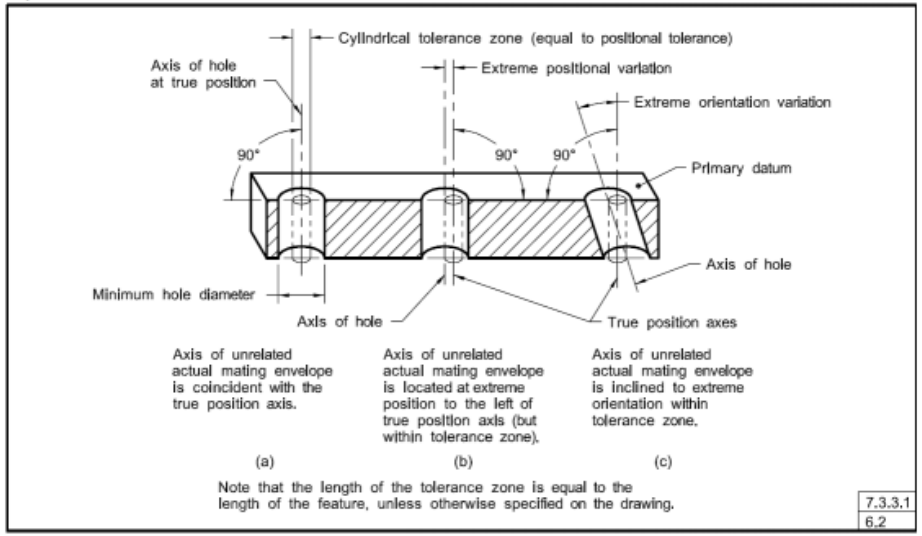
Position (Location)

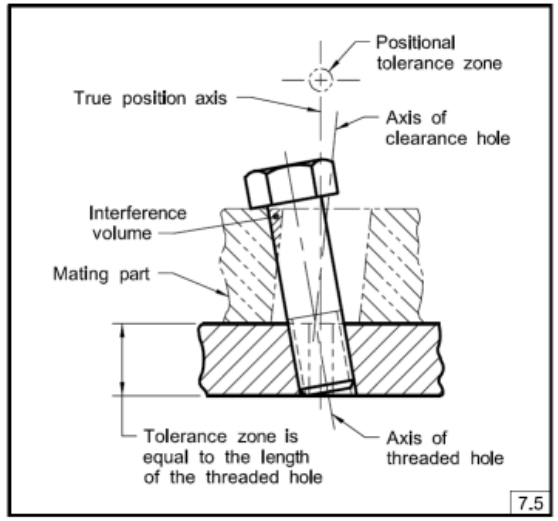


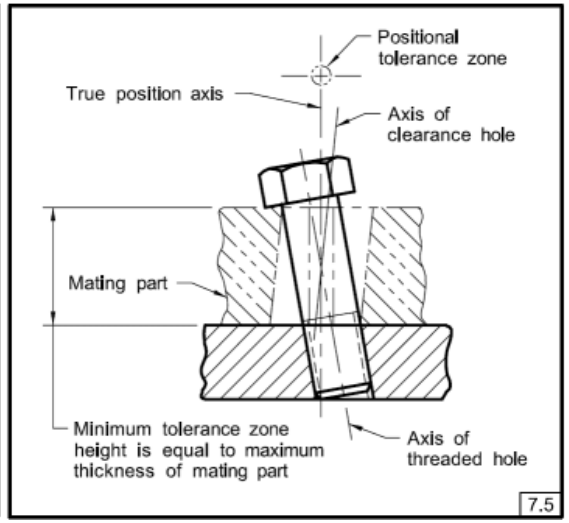








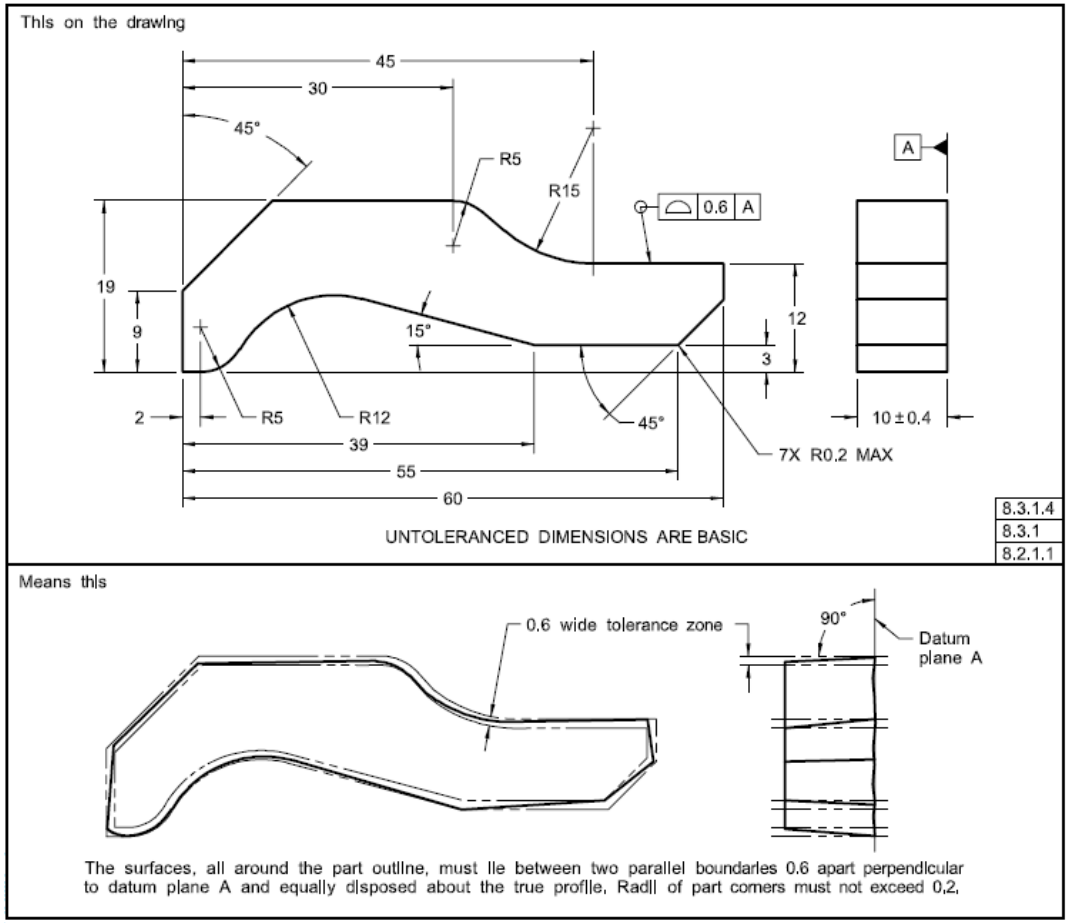


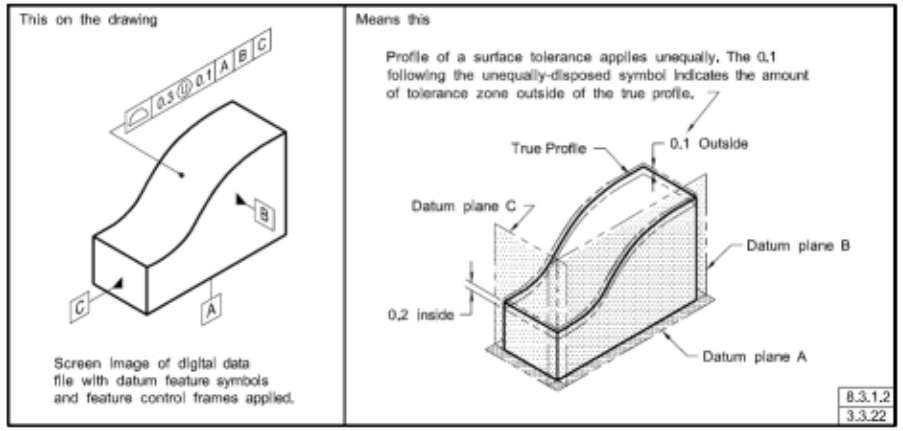


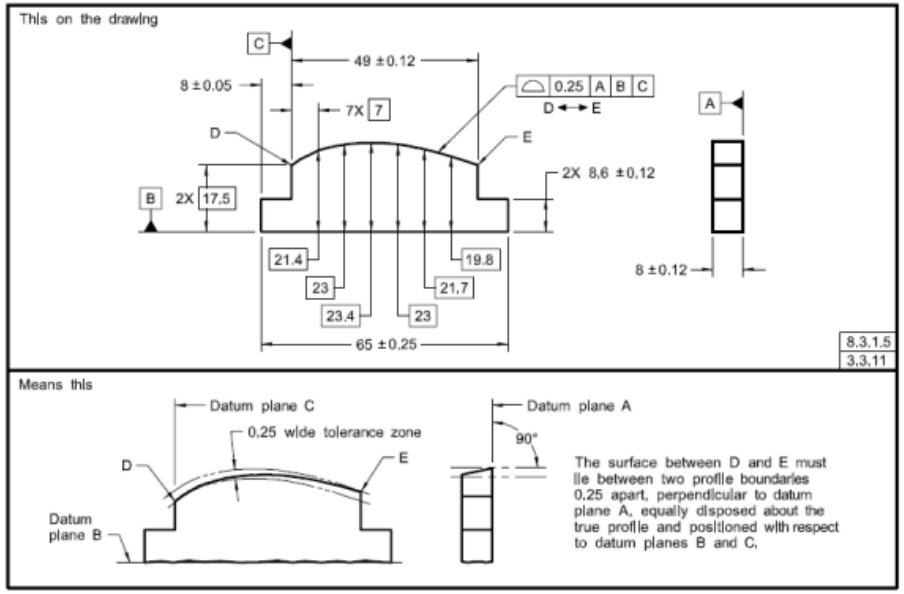
Formulas for positional tolerances.

Fixed: $T = (H - F)/2$ Floating: $T = H - F$

Profile of a Surface







Unequally disposed profile tolerance always adds material.

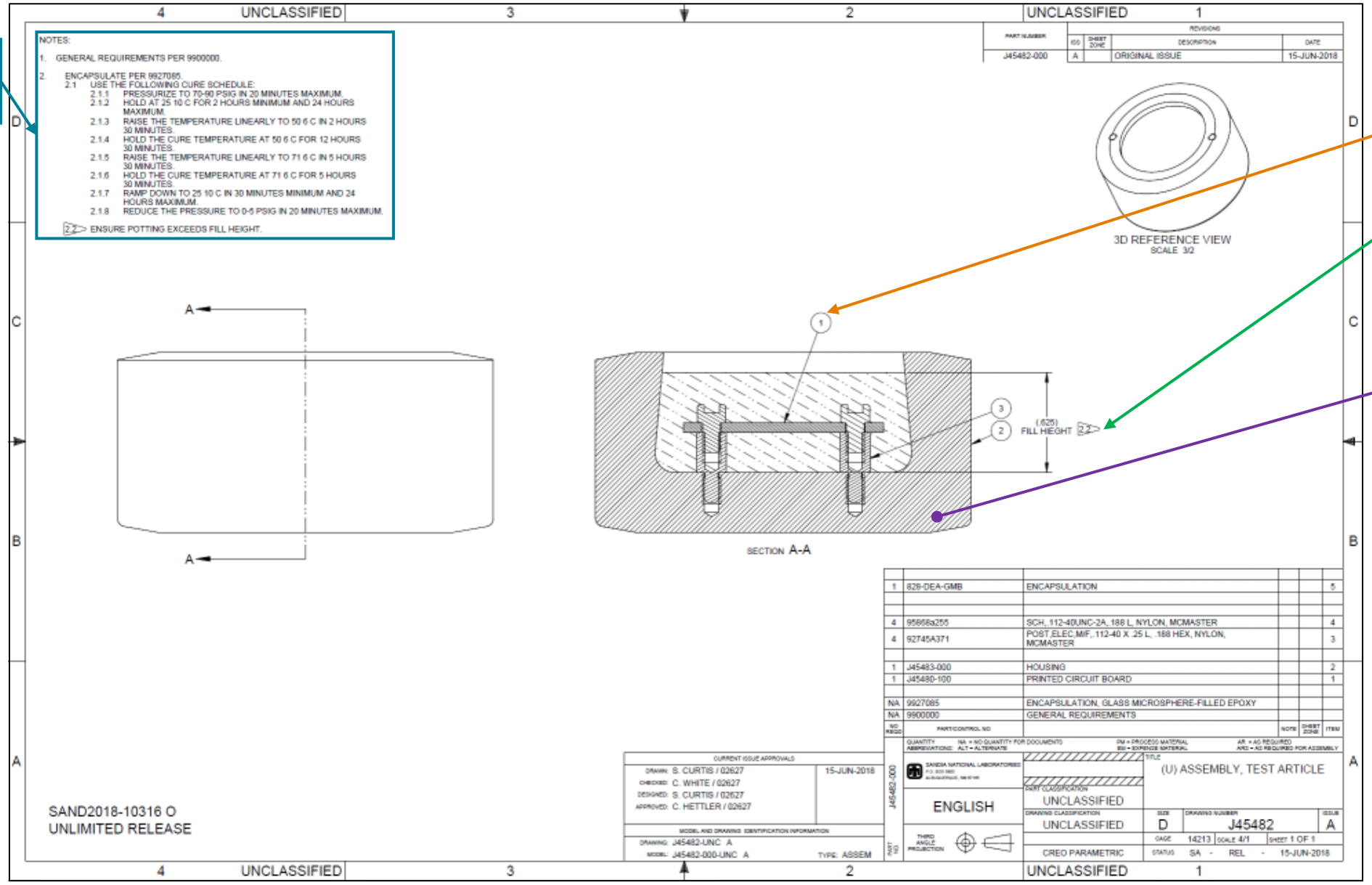
Assembly Drawing

Assembly Instructions

BOM Balloons

Flag notes

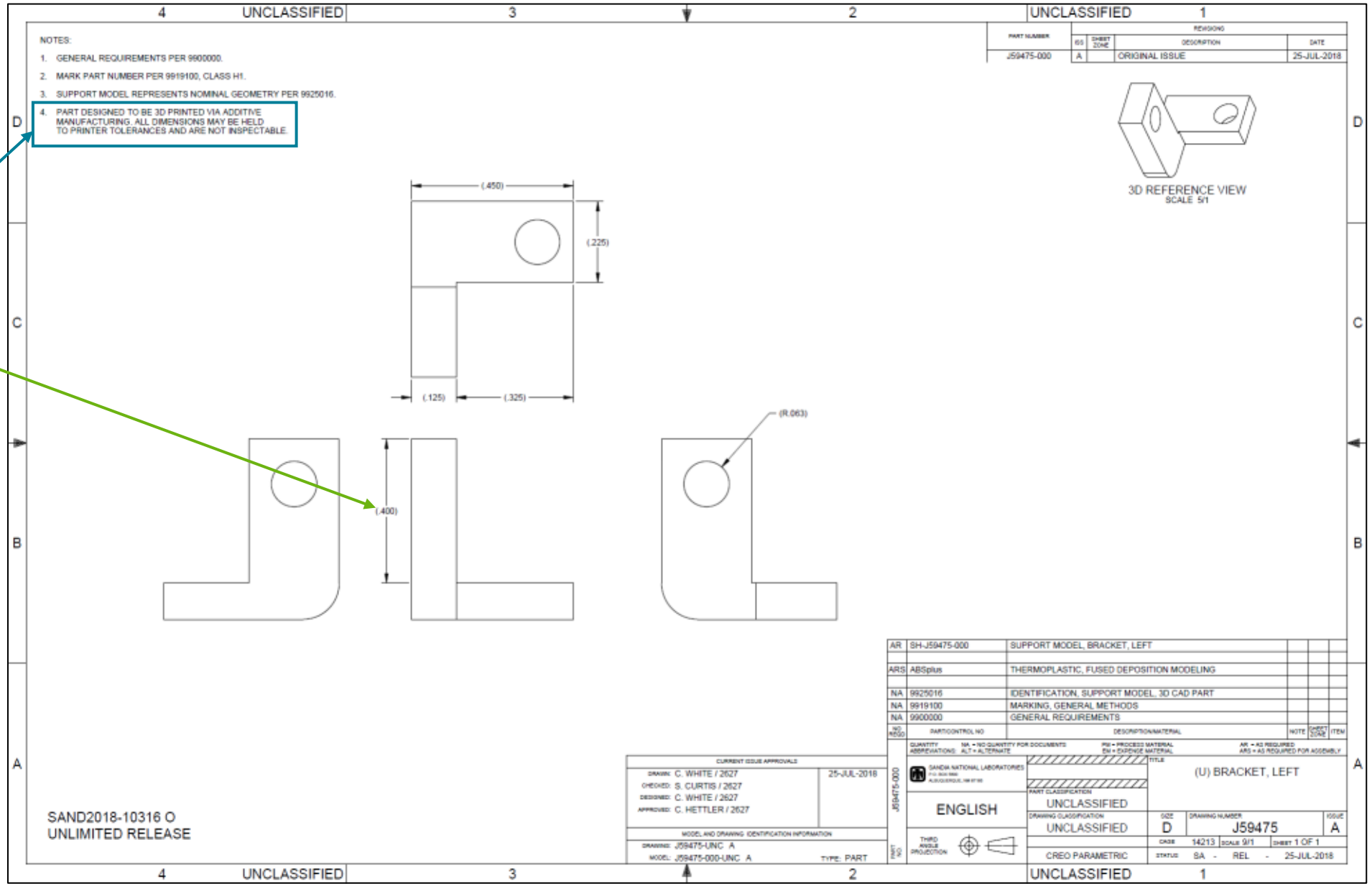
Section, detail views. Never explode.



3D Print Drawing

Do not inspect tolerances

Reference Dimensions



References

- ASME Y14.5-2009, Dimensioning and Tolerancing
- Pat McQuistion, Geometric Dimensioning and Tolerancing (GD&T)
- Don Day, The GD&T Hierarchy Y14.5-2009, Tec-Ease Inc.

Questions?