GEOMETRIC TOLERANCES AND TEXTURE

Introduction

Geometric dimensioning and tolerancing (GD&T) is an accurate technique for defining the shape, form, and relationship of features on a component. Form refers to 3D appearance and shape refers to 2D section appearance. GD&T is an advanced form of dimensioning and tolerancing. Before GD&T was developed, geometric tolerances were specified with local notes. When geometric tolerances are not explicitly specified, then size tolerances are used to ensure proper form of components. However, the control of geometric features with size tolerances is not cost effective. GD&T was initiated in the late 1950s as a technique to define geometric features more accurately on components. Its unique approach to dimensioning is the independent specification of size and form tolerances. This allows more freedom in specifying tolerances. Geometric tolerances include straightness, flatness, circularity and Cylindricity and orientations of shapes such as angularity, parallelism and perpendicularity. Run-out relates to radial features and reference axes and features that are 90^o to a reference axis. ASME (ANSI) Y14.5M is the standard for dimensioning practice in the USA.

Datum: Ideal and Real

A datum is a reference for manufacture, measurement or inspection. It may be a point, line or plane. It has an exact form or theoretical perfect form. Three datum planes or plane features are possible, namely A, B and C. Datum A is primary reference and requires at least three points for definition. Datum B is secondary datum and requires datum A and at least two points for definition. Datum C is tertiary datum and requires datum A, datum B and at least one point for definition. Fig. 1 illustrates the definitions of datum planes A, B and C and other references.

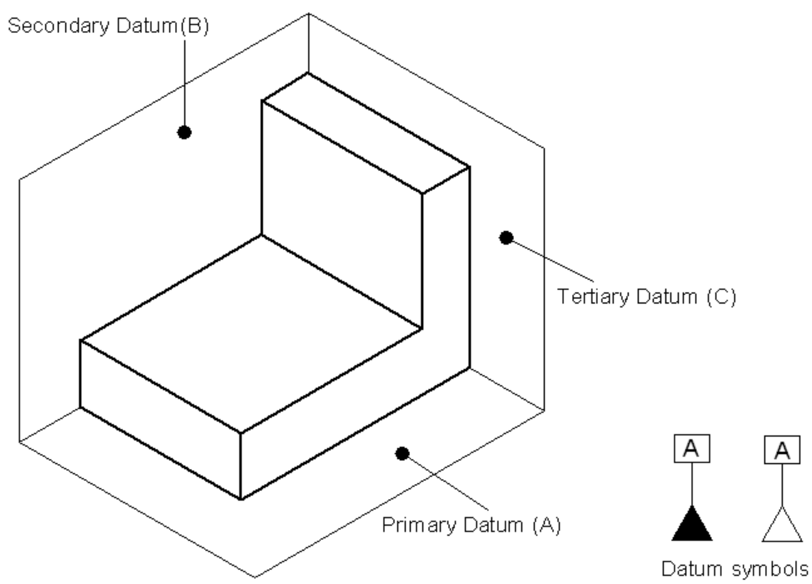


Fig. 1 Datum planes and symbols

An ideal datum is defined in space and it is theoretical but a real datum is defined on an object and is called a datum feature. A datum feature is a vertex, edge or face located on an actual drawing. A datum feature may be a point, axis, plane, or cylindrical surface, groove, recess, tab, etc. Letters are used to represent datum features but the letters I, O and Q are not allowed. Datum targets are points or area where tooling and component makes contact. These may be uneven surfaces that are common with castings and forgings. A second datum in a datum frame requires two datum targets for specification. A datum target point is indicated on a drawing by a 45^o cross (X). When a datum target area is circular, the area diameter is specified in the upper half of the datum target symbol. If a datum target is on a hidden view, the leader is shown as a hidden line. A datum frame is a system of three datum planes at right angles.

Geometric Tolerance Types and Symbols

Standard symbols are used to represent elements of geometric tolerances, datum, features, and modifiers. Table 1 summarizes the symbols and types of geometric tolerances.

Table 1: Geometric tolerance symbols

Geometric Tolerance			
Application type	Tolerance Type	Symbol	Description
Individual Features	Form	⏤	Straightness
		⏥	Flatness
		○	Circularity or Roundness
		⌭	Cylindricity
Individual/Related Features	Profile	⌒	Line profile
		⌓	Surface profile
Related Features	Orientation	∠	Angularity
		⊥	Perpendicularity
		//	Parallelism
	Location	⌖	Position
		◎	Concentricity
	Runout	↗	Circular runout
		⌰	Total runout

Form Tolerances

Straightness: straightness tolerance is the allowed maximum deviation of line element or axis from a perfect or theoretical straight line. It measures the relative height between the lowest and highest points on a surface. This tolerance applies to an edge, axis of a revolution, or line on a surface.

Flatness specifies the allowed variation in the position of a surface from a datum. It has a tolerance zone of a rectangle defined by two parallel planes separated by the tolerance value.

Circularity is the allowed maximum deviation of a circle from a perfect circle. It measures the roundness of the cross-section of a part. This tolerance applies to a surface of revolution such as cylinder, cone or sphere. The limits are two concentric circles.

Cylindricity: Cylindricity is the allowed maximum deviation of a cylinder from a perfect cylinder. It measures the straightness of the surface of a cylinder along the axis of a part. The limits are two concentric cylinders and the tolerance zone is the radial separation between the cylinders.

Orientation Tolerances

Parallelism: Parallelism defines the allowed variation of a surface, line or axis from a datum. It is the allowed maximum deviation of points on a plane from the datum.

Perpendicularity: Perpendicularity defines the angular tolerance for squareness. It relates flatness of a surface or straightness of a line inclined at 90^o to a datum.

Angularity: Angularity defines the allowed variation in angular location from a datum. It deals with the inclination of a line or surface to a datum.

Profile Tolerances

Line profile: line profile tolerance is the allowed maximum deviation of profiles of irregular shapes from the true or perfect profile. The allowed variation may be unilateral or bilateral.

Surface profile: Surface profile tolerance is the allowed maximum deviation of an outline from the true one.

Location Tolerances
Position: position tolerance is the allowed maximum deviation of position from it true location.
Concentricty: concentricity tolerance is the allowed maximum deviation of the centerline of a rounded feature from a datum.
Symmetry: symmetry tolerance is the allowed maximum deviation of a feature from a datum. It is the allowed error by which opposite sides of a feature are unequal spaced from the center plane of the datum.

Runout Tolerances
Runout tolerances deal with deviations from a perfect form when an object rotates about an axis.
Circular runout: Circular runout determines if a circular cross-section exceeds the specified tolerance. It is measured by rotating and object through a revolution about its axis. Wobble on a plane perpendicular to the axis of an object can be measured by this technique.
Total runout: Total runout tolerances are measured for all circular and profile positions as a part rotates through a revolution. Along the axis of a part, it measures the cumulative error of circularity, concentricity, straightness, angularity, taper and profile of a surface. On a perpendicular plane to the axis of a component, it measures variations in flatness and perpendicularity.

Tolerance Zone is the name of the area that is defined by the total permissible error of a geometric tolerance. Common tolerance zones are circles, rings, cylinders and rectangles.

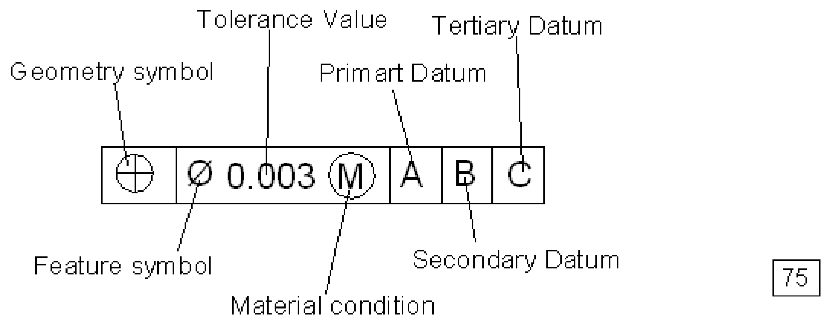
Material Conditions and Datum Modifiers
MMC: Stands for Maximum Material Condition and describes when mating components have maximum amount of material. Occurs when external component is at upper limits and internal component is at lower limits. Minimum clearance or allowance occurs at MMC of mating parts.
LMC means Least Material Condition and describes when mating components have minimum amount of material. Occurs when external component is at lower limits and internal component is at upper limits. Maximum clearance occurs at LMC of mating parts.

Table 2: Feature modifier symbols

Symbol	Meaning
Ⓜ	At Maximum Material Condition
Ⓛ	At Least Material Condition
Ⓟ	Projected Tolerance Zone
Ⓢ	Regardless of Feature Size

RFS means regardless of feature size. This indicates that tolerances apply to a geometric feature regardless of its size in the range of LMC to MMC. RFS is implied or assumed for all geometric attributes unless stated otherwise.
VC means Virtual Condition. This describes the combined effects of MMC and geometric tolerance build up. This gives the absolute maximum size for a shaft or absolute minimum size for a hole.

Feature Control Frame (FCF)
A feature control is a rectangular block with an arrangement of symbols and real numbers as shown in Fig. 2a. It is used to relate a geometric tolerance to a component feature. A leader is used to link the FCF to the feature being controlled on the component when it is not practical to attach it to an extension line. The FCF can be attached to a feature by dimension lines. However, centerlines cannot be used as links between FCF and a geometric feature. The elements on a FCF are geometric characteristics, tolerance, tolerance modifier, datum and datum modifier. The lower left portion of Table 1 shows a FCF and its interpretation.



a) Example of Feature Control Frame b) Basic dimension

Fig. 2: Feature control frame and basic dimension

Basic Dimension

Basic dimensions are sizes that are not allowed to deviate from the design size. A basic dimension is placed inside a rectangular block as shown in Fig. 2b.

Geometric Tolerancing and Dimensioning Examples

Geometric Dimensioning and Tolerancing (GD&T) is an accurate technique for defining and controlling the size of forms and shapes of features. Its unique approach in dimensioning is the independent specification of the size and geometric tolerances. It combines general and geometric tolerances by placing the feature control frame with the geometric tolerance information by a size toleranced dimension. Fig. 3 shows two examples of GD&T.

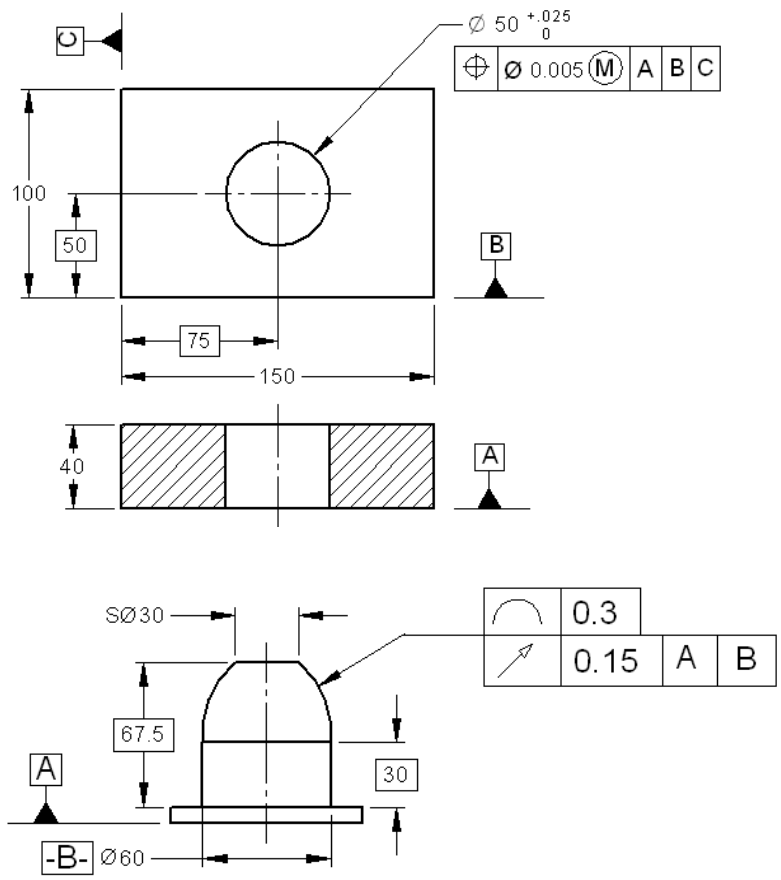


Fig. 3: GD&T examples

Surface Texture

Surface texture is used to describe several elements of the surface of a part. The major elements of surface texture at present are roughness, waviness, lays, and flaws as shown in Fig. 5. Surface roughness is the tiny irregularities on surfaces, usually of the order of microns. Surface waviness describes a more regular feature of valleys and troughs on a surface, usually of the order of millimeters. Surface roughness is superimposed on surface waviness. Lay is used to describe the direction of the predominant surface pattern. Flaw describes any recognizable defect on a surface. The control of surface roughness is important for two reasons, namely to reduce friction and control wear. These two factors influence the service life and performance quality of machines and equipments. The accuracy of measurements is related to the accuracy of the surface since fine resolution cannot be detected on rough surface. It is the responsibility of a designer to specify appropriate surface finish for functionality at minimum cost. ANSI B46.1 deals with surface control and symbols of surface texture are defined in ANSI Y14.36.

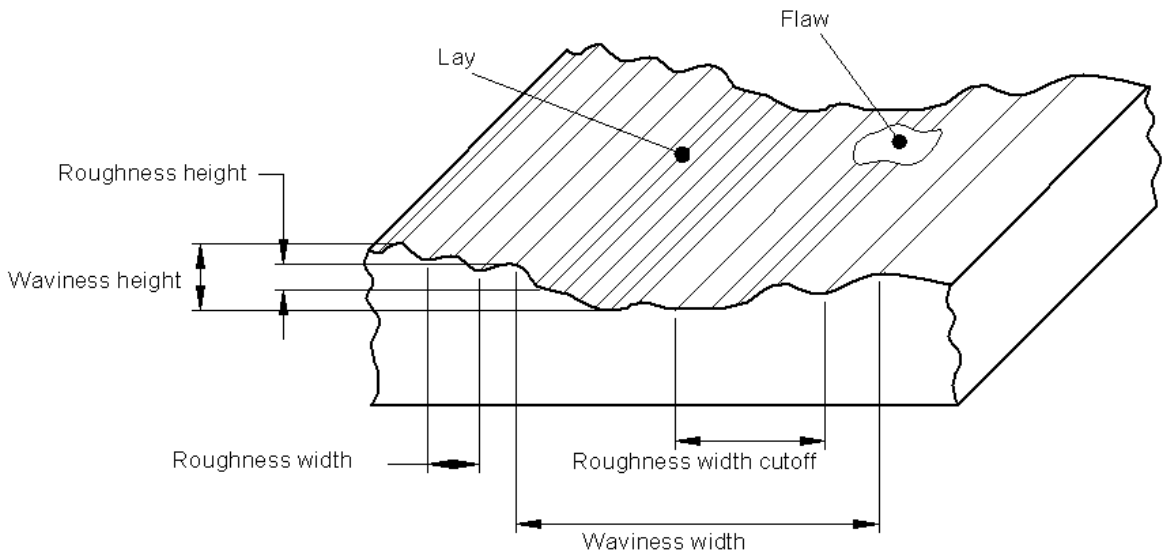


Fig. 5: Elements of surface texture

Surface Roughness Grades

Surface roughness is associated with several parameters such as average roughness height (Ra), root mean square (RMS) height (Rq), average maximum height (Rz), maximum roughness height (Rt), maximum valley depth (Rv), maximum peak height (Rp) and Motif parameter (R). These parameters are measured over a prescribed length called cut-off length. The average roughness height (Ra) appears to be the more popular parameter. It is also called the centerline average (CLA). ISO has established twelve grades (N1 to N12) of surface roughness which are shown in Table 3. This table also gives the required cut-off length. These parameters are measured in microns (SI Units) or micro-inches (English Units). The rules of thumb for surface finish are: rough turned with visible toolmarks (N10); smooth machined surface (N8); static mating surfaces or datums (N7); bearing Surfaces (N6); and fine lapped surfaces (N1).

Table 3: Surface Roughness and Cut-off Length

ISO Grade	**Surface Roughness (Ra)**		**Cut-off Length**	
	µm	µin	mm	in
N1	0.025	1	0.08	0.003
N2	0.05	2	0.25	0.01
N3	0.1	4	0.25	0.01
N4	0.2	8	0.25	0.01
N5	0.4	16	0.25	0.01
N6	0.8	32	0.8	0.03
N7	1.6	63	0.8	0.03
N8	3.2	125	2.5	0.1
N9	6.3	250	2.5	0.1
N10	12.5	500	2.5	0.1
N11	25	100	8.0	0.3
N12	50	200	8.0	0.3

Surface Texture Specification

Surface texture is generally specified with symbols as shown in Fig. 6. Surface roughness symbol generally consists of two inclined lines, one shorter than the other with an included angle of 60o, and two horizontal lines with the short one connect the inclined lines. If the short horizontal line below the average roughness range is missing in the symbol, it means machining of the surface is optional in achieving the stated roughness value. If the horizontal line is replaced by a circle, it means machining of surface is not allowed. Several parameters of surface roughness have been defined but the most popular is the arithmetic mean average surface roughness height value.

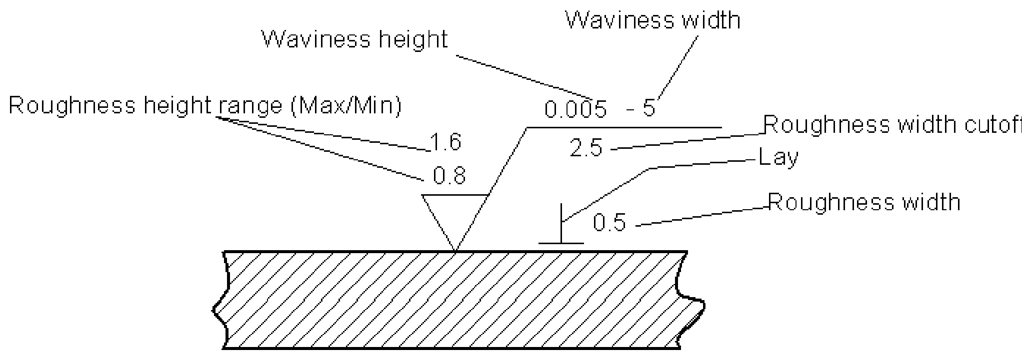


Fig. 6: Representation of surface texture

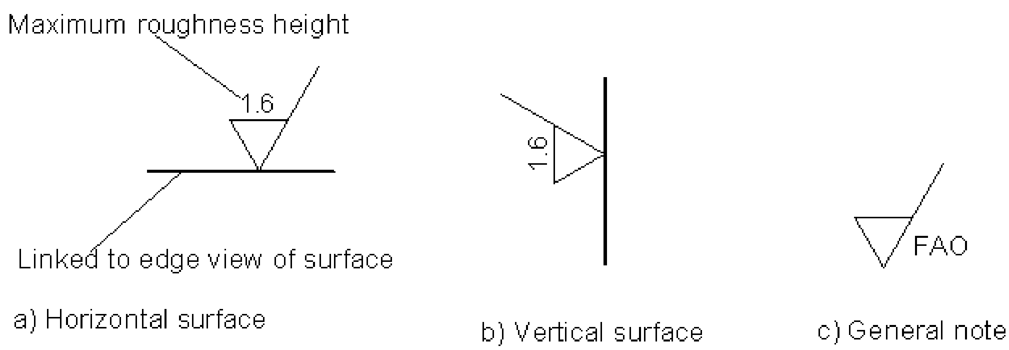


Fig. 7: Placing surface texture symbol.

Surface texture may be specified in three ways, namely full, basic, or general specification. In the full and basic specifications, the surface texture symbol should be placed perpendicular to the edge view of the surface as indicated in Fig. 6 and Fig. 7.

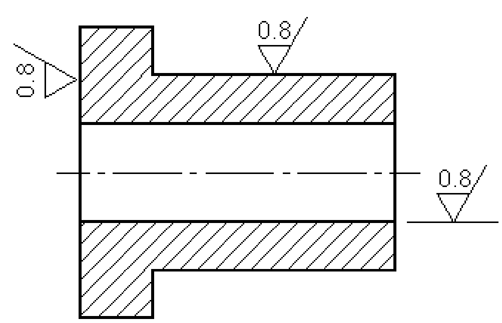


Fig. 8: Placing surface finish symbol

Full specification

All parameters of surface texture are indicated on the texture symbol for the referenced surface. Fig. 6 shows a full specification of surface texture. The labeling is only for understanding, this is not part of a specification.

Basic specification

Only important parameters of surface textures are indicated on the texture symbol for the referenced surface. How many parameters are considered important depends on the designer or engineer. In many situations, the maximum roughness height is all that is indicated in a basic specification as shown in Fig. 7 a) and Fig. 7 b).

When the maximum roughness value is indicated on a texture symbol, it implies that any value smaller than that shown is acceptable.

General specification

Parameters of surface textures are not indicated on the texture symbol for the referenced surface. The surface texture symbol and a note are added to drawing. A note such as FAO (Finish All Over) is common. Fig. 7c) is an example of a general specification. Fig. 8 is an example of a component with surface finish specification.

Component Surface Roughness Requirements

Surface roughness values vary with applications and mating conditions of components. Table 3 gives surface roughness values for average applications is some situations. Surface finish is important in high speed components due to possible fatigue failure. The recommended surface finish for precision bearing applications is 0.8 µm (32 microinches) on shafts of less than 50 mm diameter and 1.6 µm (63 microinches) for shafts of larger diameter. The suggested surface finish for bearing housing diameter is 3.2 µm (125 microinches) for all applications.

Table 3: Surface roughness for average applications

Components/Features	Surface Roughness*	
	R_a (µm)	R_a (µin)
Clearance surface (machined)	3.2 – 12.5	125 - 500
Relief areas (turned)	3.2 – 6.3	125 – 250
Mating surfaces (brackets, pads, bosses)	1.6 – 3.2	63 – 125
Tapped or die-cut threads	1.6 - 12.5	63 - 500
Surfaces for soft gaskets	1.6 - 12.5	63 - 500
Housing fits (no gaskets or seals)	1.6 – 3.1	63 - 125
Rolling surfaces-general (cams)	1.3 – 2.0	50 - 80
Slide ways and gibs, sliding surfaces and worm gears (general)	0.8 – 3.2	32 - 125
Ground threads, gear teeth, worms	0.4 – 1.6	16 – 63
Friction surfaces, brake drums, clutch plates	0.4 – 1.6	16 - 63
Sliding surfaces - precision	0.2 – 1.6	8 - 63
Rotating surfaces – precision, fatigue loaded parts	0.2 – 0.8	8 - 32

*Adapted from Budisnki (1989) Engineering Materials: Properties and Selection

Surface Roughness Production

*D*ifferent manufacturing processes have different capabilities for producing surface texture quality. Generally, machining with heavy feeds and slow speeds result in rough surfaces or high roughness values. Machining with fine feeds and high speeds give smooth surfaces or low roughness values. Often a finish machining process is carried out after a rough machining process in order to achieve a desired surface finish. Table 4 summarizes typical roughness height values for some manufacturing processes. Higher or lower roughness values may be obtained under special conditions.

Table 4: Typical surface roughness height for some manufacturing processes

Roughness height (Ra, µm)	**Manufacturing Process(es)**
12.5 – 1.6	Planning, shaping
6.3 – 1.6	Drilling, milling
6.3 - 0.4	Boring, turning
3.2 - 0.8	Broaching, Reaming
1.6 – 0.1	Grinding
0.8 - 0.1	Honing
0.4 - 0.1	Lapping
0.2 – 0.025	Supperfinishing

Surface Roughness Table - Shows the roughness average for different manufacturing processes in micrometers and microinches. The values are shown with a typical range and a less frequent range for each manufacturing process.

■	Average Range
░	Less Frequent Range

Manufacturing Process	Roughness Average Top Number - Micrometers Bottom Number - (Microinches)												
	50 (2000)	25 (1000)	12.5 (500)	6.3 (250)	3.2 (125)	1.6 (63)	0.80 (32)	0.40 (16)	0.20 (8)	0.10 (4)	0.05 (2)	0.025 (1)	0.012 (.5)
Flame Cutting	□░	░■	■░	░□	□□	□□	□□	□□	□□	□□	□□	□□	□□
Snagging	□░	░■	■■	■░	░□	□□	□□	□□	□□	□□	□□	□□	□□
Sawing	□░	░■	■■	■■	■■	■░	░□	□□	□□	□□	□□	□□	□□
Planing, Shaping	□□	□░	░■	■■	■■	■░	░░	░□	□□	□□	□□	□□	□□
Drilling	□□	□□	□░	░■	■■	■░	░□	□□	□□	□□	□□	□□	□□
Chemical Milling	□□	□□	□░	░■	■■	■░	░□	□□	□□	□□	□□	□□	□□
Elect Discharge Machining	□□	□□	□░	░■	■■	■░	░□	□□	□□	□□	□□	□□	□□
Milling	□□	□░	░░	░■	■■	■■	■░	░░	░░	□□	□□	□□	□□
Broaching	□□	□□	□□	□░	░■	■■	■░	░□	□□	□□	□□	□□	□□
Reaming	□□	□□	□□	□░	░■	■■	■░	░□	□□	□□	□□	□□	□□
Electron Beam	□□	□□	□□	□■	■■	■■	■░	░░	░□	□□	□□	□□	□□
Laser	□□	□□	□□	□■	■■	■■	■░	░░	░□	□□	□□	□□	□□
Electro-Chemical	□□	□□	□░	░░	░■	■■	■■	■■	■░	░░	░□	□□	□□
Boring, Turning	□□	□░	░░	░■	■■	■■	■■	■░	░░	░░	░░	░□	□□
Barrel Finishing	□□	□□	□□	□□	□░	░░	░■	■■	■░	░░	░□	□□	□□
	50 (2000)	25 (1000)	12.5 (500)	6.3 (250)	3.2 (125)	1.6 (63)	0.80 (32)	0.40 (16)	0.20 (8)	0.10 (4)	0.05 (2)	0.025 (1)	0.012 (.5)
Electrolytic Grinding	□□	□□	□□	□□	□□	□□	□░	■■	■░	░□	□□	□□	□□
Roller Burnishing	□□	□□	□□	□□	□□	□□	□░	░■	■░	░□	□□	□□	□□
Grinding	□□	□□	□□	□░	░░	░■	■■	■■	■■	■░	░░	░□	□□
Honing	□□	□□	□□	□□	□□	□░	░■	■■	■■	■░	░░	░□	□□
Electro-Polish	□□	□□	□□	□□	□□	□░	░■	■■	■■	■░	░░	░░	░□
Polishing	□□	□□	□□	□□	□□	□□	□░	░■	■■	■░	░░	░░	░□
Lapping	□□	□□	□□	□□	□□	□□	□░	░■	■■	■■	■░	░░	░□
Super Finishing	□□	□□	□□	□□	□□	□□	□░	░░	░■	■■	■■	■░	░□
Sand Casting	□░	░■	■░	░□	□□	□□	□□	□□	□□	□□	□□	□□	□□
Hot Rolling	□░	░■	■░	░□	□□	□□	□□	□□	□□	□□	□□	□□	□□
Forging	□□	□░	░■	■■	■░	░□	□□	□□	□□	□□	□□	□□	□□
Permanent Mold Casting	□□	□□	□□	□░	░■	■░	░□	□□	□□	□□	□□	□□	□□
Investment Casting	□□	□□	□□	□░	░■	■░	░░	░□	□□	□□	□□	□□	□□
Extruding	□□	□□	□░	░░	░■	■■	■░	░□	□□	□□	□□	□□	□□
Cold Rolling, Drawing	□□	□□	□□	□░	░■	■■	■░	░░	░□	□□	□□	□□	□□
Die Casting	□□	□□	□□	□□	□░	░■	■░	░□	□□	□□	□□	□□	□□
	50 (2000)	25 (1000)	12.5 (500)	6.3 (250)	3.2 (125)	1.6 (63)	0.80 (32)	0.40 (16)	0.20 (8)	0.10 (4)	0.05 (2)	0.025 (1)	0.012 (.5)

http://www.engineershandbook.com/Tables/surfaceroughness.htm (2-20-12)

ISO 1302
http://www.bcmac.com/pdf_files/surface%20finish%20101.pdf

Surface Texture

Surface Texture Symbols, ASME/ANSI Y14.36M-1996, defines the American National Standard symbology for expressing desired surface finishes resulting from processing solid materials. Qualitative values can be expressed for roughness, waviness and lay. The Y14.36M standard does not address the manufacturing processes required to produce a particular surface texture value, nor how it is verified. Surface texture definitions and texture verification methods are shown in ASME/ANSI B46.1-1995, Surface Texture (Surface Roughness, Waviness and Lay).

The use of surface texture symbology is optional. Where no surface texture symbology is used, the surface finish may be considered satisfactory if it is produced within the limits of size and form in accordance with ASME/ANSI Y14.5M-1994. When surface texture symbology is used, it is considered applicable to the finished surface of the object.

ISO Surface Finish Designations

ISO standards are published in a different fashion than American National standards in that they are written as smaller focused documents (often called parts), which in turn reference other standards or other parts of the same standard. In the case of surface finish, numerous individual standards taken as a whole form a set of standards roughly comparable in scope to American National Standard ANSI/ASME Y14.36M (See Section 11.2 for complete coverage on ANSI/ASME Y14.36M). To add to the confusion of U.S. users, and unlike the numbering scheme used by ANSI, ISO standards on a particular topic often do not carry sequential numbers, nor are they in consecutive series.

The primary ISO standard dealing with surface finish, ISO 1302:1992 is concerned with the methods of specifying surface texture symbology and additional indications on engineering drawings. This and all ISO standards are expressed in SI metric units, with commas (,) used as decimal points. Other ISO standards are referenced for constituent provisions, but not directly discussed in the ISO 1302 standard. For instance:

ISO 468:1982 Surface roughness � Parameters: Their values and general rules for specifying requirements.

ISO 4287:1997 Surface texture: Profile method � Terms, definitions and surface texture parameters.

ISO 4288:1996 Surface texture: Profile method � Rules and procedures for the assessment of surface texture.

ISO 8785:1998 Surface imperfections � Terms, definitions and parameters.

ISO 10135-1:CD Representation of parts produced by shaping processes � Part 1: Molded parts.