TRANSFERRING 3D PARTS TO DRAFTING DETAILING

Aylin YENİLMEZ GÜRKÖK

TRANSFERRING 3D PARTS TO DRAFTING DETAILING

- Geometry (Topology)
- Dimensions
- Tolerances for each dimension
- Geometric Tolerances of Features (Datums, GD&T, feature control frames)
- Manufacturing Information (Surface finish, welding notation, etc.)
- Inspection information
- Assembly instructions
- Product information *Materials, Suppliers, Part numbers, Bill of materials, Revision history, etc.*

Geometry (Topology)

- **Geometry** The word *geometry* is Greek for *geos* - meaning earth and *metron* - meaning measure. Geometry was extremely important to ancient societies and was used for surveying, astronomy, navigation, and building. Geometry, as we know it is actually known as Euclidean geometry which was written well over 2000 years ago in Ancient Greece by Euclid, Pythagoras, Thales, Plato and Aristotle just to mention a few. Geometry is the study of angles and triangles, perimeter, area and volume.
- **Terms** *(Undefined)*
- **Point**
- **Line**
  Terms *(Defined)*
- **Line Segment**
- **Ray**
- **Angle**
- **Plane**

- **Topology** (Greek topos, place and logos, study) is a branch of mathematics, an extension of geometry. Topology begins with a consideration of the nature of space, investigating both its fine structure and its global structure. Topology builds on set theory, considering both sets of points and families of sets.
- The word topology is used both for the area of study, and for a family of sets with certain properties described below. Of particular importance in the study of topology are functions or maps that are continuous. These functions stretch space without tearing it apart or sticking distinct parts together.
- **topology**' **means** `the way things are connected together'

Tolerances for each dimension

- (In tree display) Annotations>Show Feature Dimensions (right click)
- Click on the dimension>Tolerance/Precision>Bilateral> …

Geometric Tolerances of Features (Datums, GD&T (Geometric Dimensioning & Tolerancing), feature control frames)

- Understanding the international engineering language of Geometric Dimensioning & Tolerancing (GD&T) is essential for communicating in today's highly competitive global marketplace. This program builds the ability to read and interpret GD&T symbols, which provide increased information about the function and relationship of part features. One critical result is the proper fit of mating parts.

Manufacturing Information (Geometric Tolerances)

Symbol	Description	Geometry
	ANGULARITY	ORIENTATION
	CONCENTRICITY	LOCATION
	CYLINDRICITY	FORM
	FLATNESS	FORM
	PARALLELISM	ORIENTATION
	PERPENDICULARITY	ORIENTATION
	POSITION	LOCATION
	PROFILE	PROFILE
	PROFILE OF A LINE	PROFILE
	CIRCULARITY	FORM
	RUNOUT	RUNOUT
	STRAIGHTNESS	FORM
	SYMMETRY	LOCATION
	TOTAL RUNOUT	RUNOUT

Symbol	Modifier
Ⓕ	FREE STATE
Ⓛ	LEAST MATERIAL CONDITION
Ⓜ	MAXIMUM MATERIAL CONDITION
Ⓟ	PROJECTED TOLERANCE ZONE
Ⓢ	REGARDLESS OF FEATURE SIZE
Ⓣ	TANGENT PLANE
Ⓤ	UNILATERAL

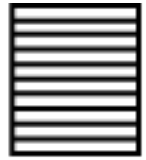
Geometric tolerances specify the maximum variation that is allowed in form or position from true geometry. The tolerances of form control straightness, flatness, parallelism, angular displacement etc.

Manufacturing Information (Surface Finishing)

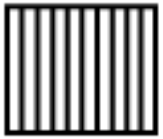
- Surface finishes are usually specified with a "check mark" on the blueprint as shown in the in graphic below. Surface finishes are specified in micro inches and are located on the left side of the symbol above the check mark "V".
- The waviness requirement (if specified) is usually given in thousands of an inch and is located on the top right of the symbol.
- The roughness width requirement (if specified) is usually given in thousands of an inch and is located on the bottom right of the symbol.
- The lay direction requirement (if specified) is usually represented by a symbol and is located right below the roughness width requirement.

Manufacturing Information (Surface Finishing)

//	which means a parallel lay
⊥	which means a perpendicular lay
C	which means a circular lay
R	which means a radial lay
M	which means a multi-directional lay
X	which means an angular lay in both directions

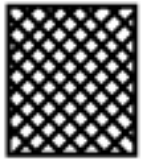


Parallel



Perpendicular

Circular



Angular

Multidirectional

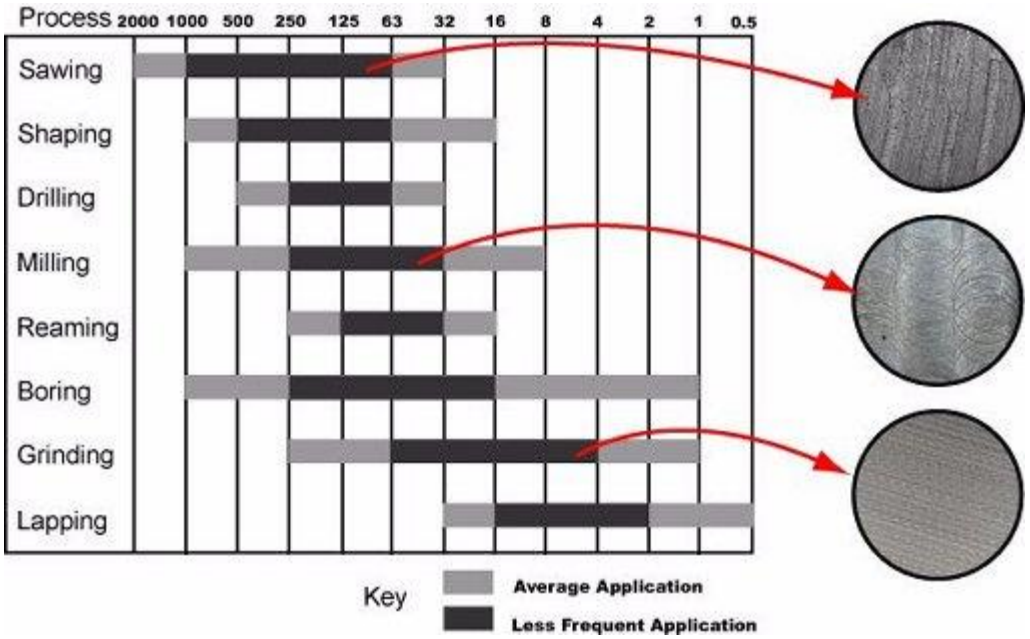
Radial

Surface Roughness

- Maximum roughness
- Minimum roughness
- Material removal allowance
- Production method/treatment
- Sampling Length
- Other roughness values
- Roughness spacing
- Lay direction

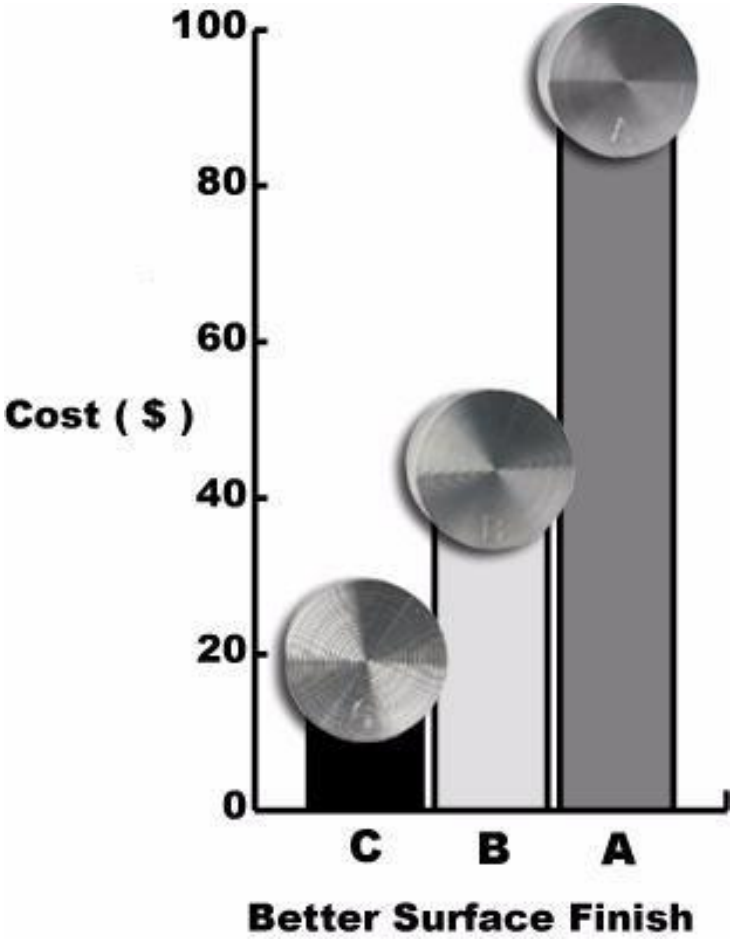
Production method/treatment

- As mentioned, different machining process produce different finishes. This graphic shows what you can expect from the typical machine shop processes.



Typically the larger the radius, the better the surface finish.
As a tool nose moves away from pointed toward rounded the depth of the profile decreases, producing a better finish.
The type of tooling used for any machining process can better the surface finish if care is taken.

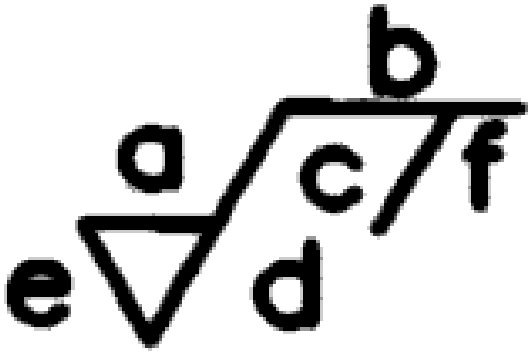
Production method/treatment



- High surface finish is not free. There are limits on machine capabilities and productions rates that can increase the cost of finish. For example an inexpensive lathe can not produce a surface finish of Ra=2. To achieve that finish the machining will have to performed on a much more expensive cylindrical grinder. Also the feed-rate to achieve a high finish is much slower than normal and cost time which adds cost. The graphic gives a simple example.

Surface Roughness Symbol

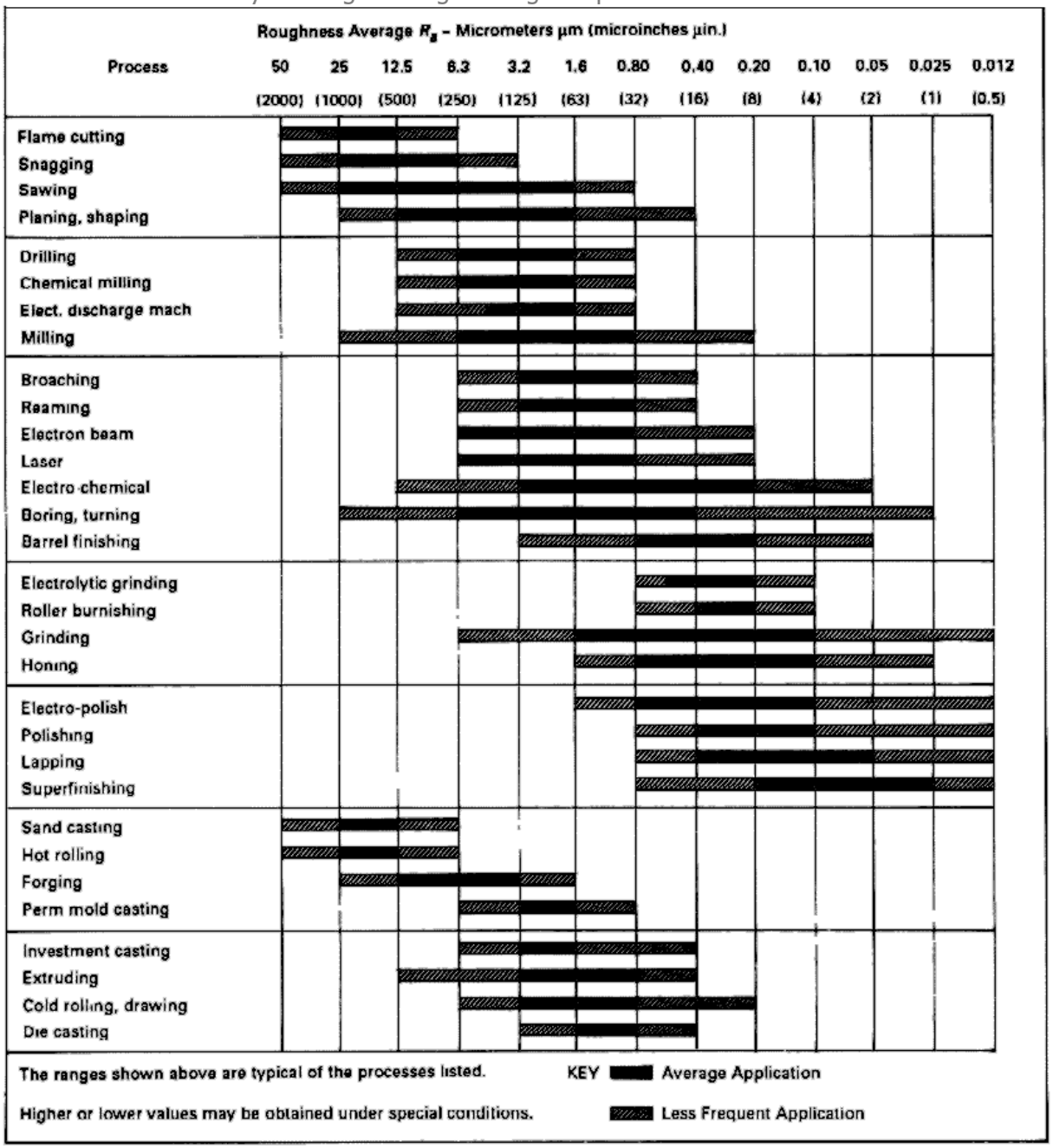
- a – roughness value (Ra)
- b – production method, treatment, coating, etc.
- c – sampling length (cutoff)
- d – direction of lay
- e – minimum material removal
- f – roughness value other than Ra



Surface finish roughness grade number

Roughness values Ra		Roughness Grade Numbers
μm	μin	
50	2000	N12
25	1000	N11
12.5	500	N10
6.3	250	N9
3.2	125	N8
1.6	63	N7
0.8	32	N6
0.4	16	N5
0.2	8	N4
0.1	4	N3
0.05	2	N2
0.025	1	N1

Surface finish may be denoted by a roughness grade number. Here is a table that specifies the Ra values for roughness grade numbers. (ISO standard – 1992)


Roughness Average R_a – Micrometers μm (microinches μin.)
Process
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 (0.5)
Flame cutting
Snagging
Sawing
Planing, shaping
Drilling
Chemical milling
Elect. discharge mach
Milling
Broaching
Reaming
Electron beam
Laser
Electro-chemical
Boring, turning
Barrel finishing
Electrolytic grinding
Roller burnishing
Grinding
Honing
Electro-polish
Polishing
Lapping
Superfinishing
Sand casting
Hot rolling
Forging
Perm mold casting
Investment casting
Extruding
Cold rolling, drawing
Die casting
The ranges shown above are typical of the processes listed.
Higher or lower values may be obtained under special conditions.
KEY
Average Application
Less Frequent Application

Creating 3D Model and Transferring to 2D Model

- Front, Top and Left View
- Model Items
- Source>Entire Model
- Dimensions>Select All
- Annotations>Select All
- Dimension>Witness/Leader Display>Inside (for arrows)
- (Dimension Fonts) Tools>Options>Document Properties>Annotations Font>Dimension>Height>12 Points
- Save the drawing file as a DWG file

From 2D AutoCAD file to 3D Solidworks file

- Open the file>import a new part
- Open 2D to 3D menu
- Front, Top, and Left View
- Align them if necessary
- Extrude boss or cut
- 2D to 3D.avi (you can download the file from the webpage)

From 3D to 2D

- Create your own 3D model with dimensions
- Convert it to a drawing (change the title, name and anything you need)
- Annotations>Show Feature Dimensions
- Complete dimensions
- Make drawing from part
- Annotations>Model Items
- Rebuild
- Edit Sheet Format
- Add surface finishing symbols and tolerances that you think they are appropriate for your part

FINALLY

- Send your assignment through Blackboard
- Your drawing files as

  YourSurnameName_YourNumber_A5.sldprt
  YourSurnameName_YourNumber_A5.slddrw

and

YourSurnameName_YourNumber_A5.jpeg

(Do not forget to compress them!)