Lezione 6 - Tolerances

GD & T → Geometrical Dimensioning & Tolerances

- → Recap (Intro)
- → Analysis of linear chains of dimensional tolerances
- → Allocation of tolerances

Introduction

Manufactured parts require precision, we would to be perfect but we always have errors so we accept tolerances.

- Precision is required to ensure the functioning of a part as intended
- It is impossible to make a part economically to the right dimension, this is due to:
  - → inaccuracies of machines and tools
  - → inaccuracies in setting the work to the tool
  - → Error in measurement.

Operators therefore have to be given some allowable margin so they can produce the part.

We assume there are errors but we have to keep them under control.

Error types

- Size
  - Dimensional Tolerances.
- Geometrict
  - Macro → Geometrical tolerances
  - Micro → Surface Roughness

Dimensional Tolerances

Tolerances → permissible variation in the size of a part

Limits → the two permissible extremes between which the actual size is contained

Allowance

↳ Dimensional difference between the maximum limits between mating parts.

If the allowance is positive, it will result in minimum clearance between the mating parts, and if negative it will result in maximum interference.

→ i.e. maximum clearance or interference.

→ allowance is the prescribed difference between two mating parts.

Basic Size → Dimensione Nominale
Actual Size → " Reale
→ From design calculation
→ Size obtained after manufacturing.

Symbols (letter + number)
→ letter → grade (capital → hole, lower case → shaft) → fundamental deviation
→ number → IT grade → defines quality of tolerance.
→ where the tollerance is positioned relative to the zero line

H → minimum = 0
h → maximum = 0
} fundamental deviation is 0.

Shafts → $ei = es - IT$
Holes → $EI = ES - IT$

Clearance Fit → always have clearance
Transition Fit → either can be longer than the other
Interference fit → shaft always longer than whole.

H/h never interfere, they are exactly the same.

In clearance fit we cannot use it for clearance, so we can save a bit of many increasing the IT grade, reducing the tollerances.

In clearance fit, it isn't advantageous to use higher

IT grade since it can mean a large variation in the clearance, and since this indicates the stress, it means that the range of coupling stresses can also be very large, which we do not want. In this worst case the exercise clearance can lead to stresses which overcome our yield stress.
A large tolerance doesn't mean anything since the behaviour is the same.
Transition fits are used for coupling.

<! Table of classes of fits>

For each category there are indications of typical combinations.

Different processes have tolerance grade ranges.

A lower grade, means higher precision and so lower cost.

Holes typically have higher grades since it's easier to work on a shaft rather than a holes. So shafts are typically one grade better.

Geometrical Tolerances (Tolerances on form and position)

- → Form Variation → variation on form feature (e.g. circularity, flatness, etc.)
- → Position Variation
  - ↳ Variation in the actual position of the form feature relative to the ideal position.

Geometric Tolerances

We don't have tolerance ranges, we have tolerance zones, these are imaginary areas within which we want to stay.

E.g we accept if the actual profile exists within two concentric circles.

or if a plane exists within two parallel planes.

There are different types of tolerances

Some constraints require a refence.

Run-out (oscillazione semplice o totale)

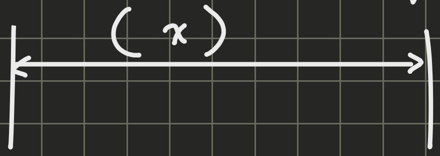
radial displacement on measurement in one circle of the profile.

Circularity is different because we make a point cloud and see if the shiele is within two concentric circles.

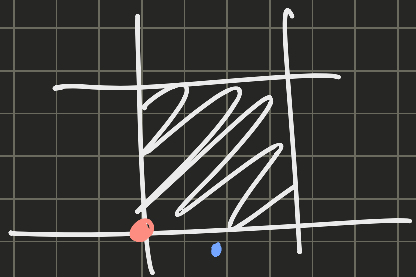
Run-out is relative to two supporting surfaces, it sums up multiple errors, the error in the measure, the error in two reference surfaces and the eccentricity.

Geometric tolerances are becoming more useful, and might replace dimensional tolerances.

A dimension between parenthesis, is auxiliary, it can only be used for information, not to provide specific indication.



There is ambiguity in the possible position of the holes.



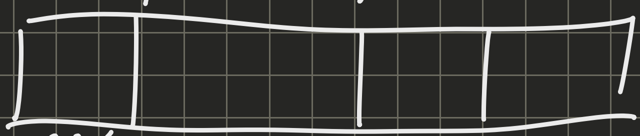
There is no reason for •, to not be acceptable when • is.
• is closer to the ideal position, relative to •.

This is why using geometrical tolerances can be useful rather than using dimensional tolerances.

We use basic dimensions, then use geometric tolerances to define a tollerance.

4 x ϕ20 ± 0,1

Geometric tolerance



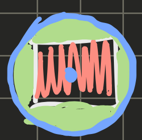
→ This allows to use some dimensional tolerances.

Symbol — Bore feature and characteristic section. — Datum

We can use the reference planes, and dimension the distance from them.

$.28 = .1 \times 2 \times \sqrt{2}$

.1 I



We have defined a cylinder within which we accept the center position of the hole, through the use of geometric tolerances we have less ambiguous designs and a greater range of acceptable values.

We cannot continued dimensional and geometric tolerance, for the whole drawing.

Hole when we are interested in the internal dimension, shaft when external

Tolerance analysis of linear dimensional chains

G_{max} = Max hole - minimum of all shaft

G_{min} = Min hole - max of all shafts

G_{nom} = Nom hole - Nom shafts

$T_G = G_{max} - G_{min} = T_A + T_B + T_C + T_D + T_E$

Tollerance is the sum of serie of tolerance dimensions.

Allocation of Tolerances.

↳ if we have desired tolerance and have to assign to others to get a tolerance somewhere.

Some things are bought, so we cannot change that tolerance