Geometric Dimensioning and Tolerancing in automotive design

An assignment by Said Bashir Bello for the Gunsel Car Factory

3/6/2021

Table of contents

Introduction

Metrology can be defined as the science of measurement.

Its most basic assumption is that *measured value differs from the true value.*

It has three foundational laws:

You cannot measure anything accurately.

Even if you manufacture it accurately there is no way of knowing if that is the true length.

If attempts are made to achieve perfection the cost of production increases tremendously.

Introduction

Any production process contains 3 elements

Man:

This is the most unpredictable element.

Man has many attributes which vary every day.

For example changes in mood, expertise and personnel all affect the production process.

Material:

There must be thermal expansitivity no matter how small.

Which means measured value will differ based on the temperature.

Machine:

It is an integral unit of tiny component's.

Due to the first law, since a machine is made up of tiny components when they are integrated, they cannot manufacture anything accurately.

Introduction

Due to the inaccuracies caused by the factors mentioned above, we have to find some understanding between the designers and manufacturers.

Some permissible variations in the dimensions are therefore needed.

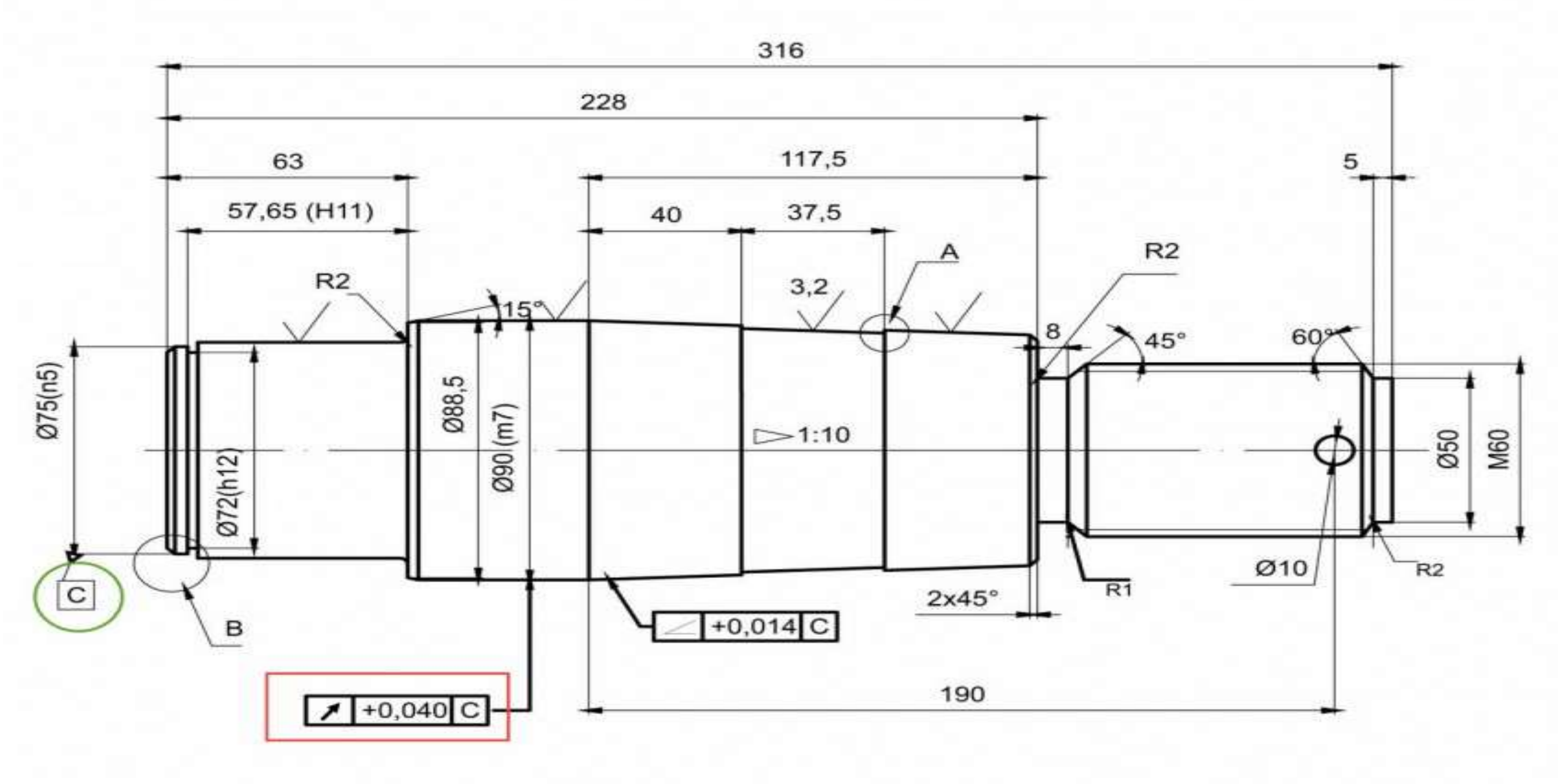
This gave rise to two concepts :

LIMITS: The dimensions of a manufactured part can be made to lie between maximum and minimum limits.

TOLERANCE: The permissible variation in size of dimension is called tolerance.

Introduction

Definition: Geometric Dimensioning and Tolerancing, is a system for defining and communicating design intent and engineering tolerances that helps engineers and manufacturers.



Principles of Geometric Dimensioning and Tolerancing in automotive design and manufacturing

Principles

- Manufacturing work pieces without variations from the nominal form is impossible, there are always deviations in size, shape, orientation, and position in work pieces. It is essential therefore that the drawing tolerances thoroughly define the work piece, i.e. each property (size, shape, etc.), only then will the producer be able to select the most cost-effective production process.
- In order to specify geometrical tolerances, the work piece is considered to be composed of features (geometrical elements), such as planes, cylinders, cones, spheres, tori, etc.
- These features are described using various symbols, zones and points of reference, which make up the foundations geometrical dimensioning and tolerancing.

Principles

Datum:

A datum is a theoretical, ideal feature established to determine the orientation, location, and/or run-out

Types of datum:

Datum feature: A true (non-ideal) integral property of a target that is utilized to establish a datum (e.g. surface of a part or hole).

Simulated Datum Feature:

An actual surface having a sufficiently exact form that comes into touch with a datum feature and is used to establish a datum (e.g. surface plate, bearing, and mandrel).

Datum System:

A collection of datum's formed by merging more than one unique datum for usage as a reference for toleranced features.

Principles

Types of tolerance:

Form tolerance

Form tolerance is a fundamental geometric tolerance that governs the target's shape (part).

None of the form tolerance qualities require a datum—forms can be derived independently.

orientation tolerance

The orientation tolerance of a form governs its orientation in regard to a reference

It is a geometric tolerance for features connected to datum's since a datum is always required to express orientation tolerance.

Principles

Types of tolerance:

Location tolerance

The location tolerance of a feature indicates its real position in regard to a reference.

It is a geometric tolerance for features connected to datum's since a datum is always required to express location tolerance.

Run-out Tolerance

Run-out tolerance is a geometric tolerance that describes the run-out fluctuation of a target's feature as it is rotated on an axis (specified straight line).

A datum is always required to show run-out tolerance; hence, it is a geometric tolerance for datum-related properties.

principles

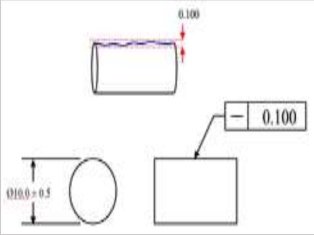
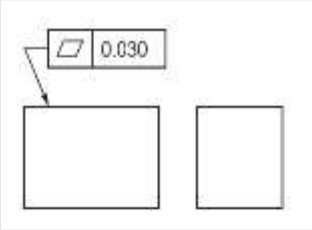
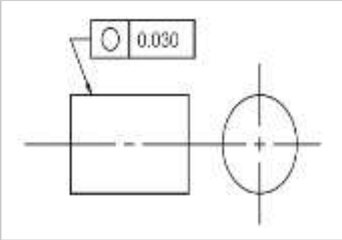
Symbols:

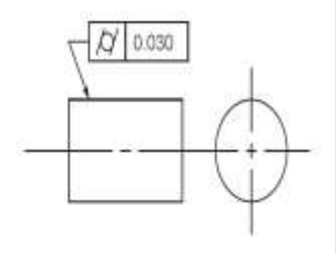
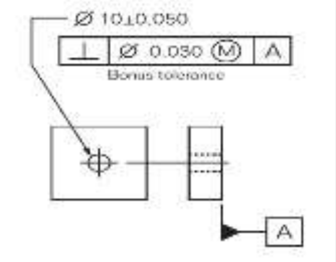
Geometric tolerances are defined on a design using symbols. We now have 16 geometric tolerance symbols, which are classified according to the tolerance they give.

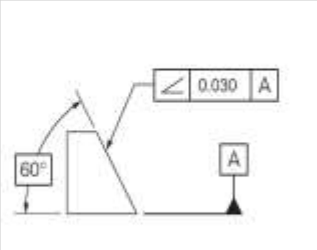
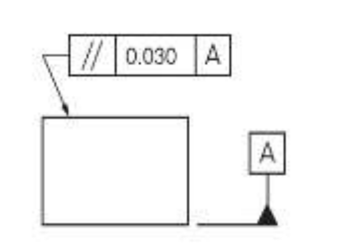
Using the GD&T detailed standards for your drawings, manufacturers all around the world can fully understand the design intent.

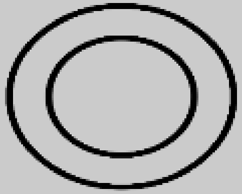
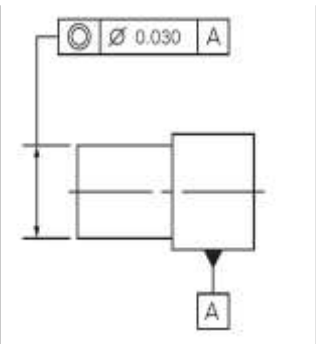
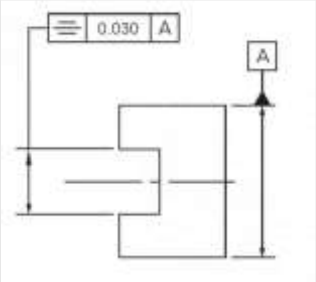
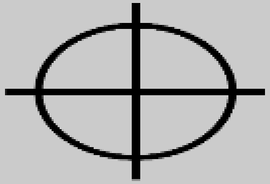
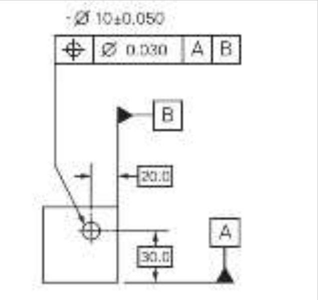
With the standard symbols, you can also create precisely defined designs that take the function of the part into account and allow for more accurate tolerancing.

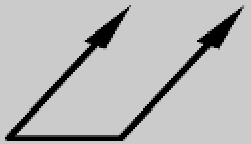
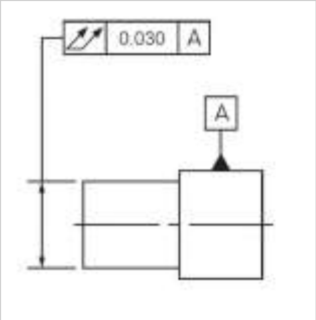
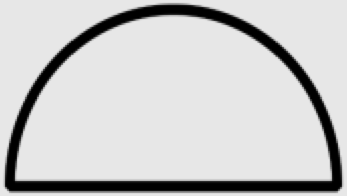
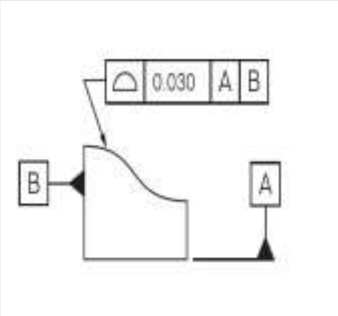
Tolerancing Symbols

CHARACTERISTIC	SYMBOL	DEFINITION	TYPE	Drawing
Straightness	—	*Straightness* is divided into line element straightness and axis straightness	**Form control**	
Flatness	▱	*Flatness* means straightness in multiple dimensions, measured between the highest and lowest points on a surface.	**Form control**	
Circularity	○	*Circularity or roundness can be described as straightness bent into a circle.*	**Form control**	

CHARACTERISTIC	SYMBOL	DEFINITION	TYPE	Drawing
Cylindricity	⌭	*Cylindricity is basically flatness bent into a barrel. It includes straightness, roundness, and taper, which makes it expensive to inspect*	**Form control**	
Perpendicularity	⊥	*Perpendicularity means flatness at 90 degrees to a datum. It specifies two perfect planes the feature plane must lie in between.*	**Orientation controls**	

CHARACTERISTIC	SYMBOL	DEFINITION	TYPE	Drawing
Angularity	∠	*Angularity is flatness at an angle to a datum and is also determined through two reference planes spaced the tolerance value apart.*	**Orientation controls**	
Parallelism	//	*Parallelism mea ns straightness at a distance.*	**Orientation controls**	

CHARACTERISTIC	SYMBOL	DEFINITION	TYPE	Drawing
Concentricity		*Concentricity compares the location of a feature axis to the datum axis*	**Location controls**	
Symmetry		*symmetry ensures that non-cylindrical parts are similar across a datum plane.*	**Location controls**	
True Position		*True position tolerance in GD &T informs us of the maximum allowable deviation of a feature (e.g. hole, slot) from its true position*	**Location controls**	

CHARACTERISTIC	SYMBOL	DEFINITION	TYPE	Drawing
Runout - Total Runout		*Total Runout is measured on multiple points of a surface, not just describing the runout of a circular feature but of an entire surface.*	**Runout controls**	
Profile a line - Profile a Surface		*Surface Profile creates through two offset surfaces between which the feature surface must fall*	**Profile controls**	

Why implement GD&T Processes to automotive design and manufacturing

The GD&T method decreases the quantity of notes, measurements, and tolerances necessary on a design when applied correctly.

GD&T provides a precise approach for providing a reference coordinate system that may be used during production and inspection operations by establishing datums.

GD&T is the system that allows developers and inspectors to optimize functionality without increasing cost.

Furthermore, when the function of the part is considered, GD&T enables for a wider tolerance range to be used, minimizing the number of functional parts that are rejected.

The most significant advantage of GD&T is that it describes the design intent rather than the actual geometry.

It is easier to define product geometry in terms of its intended functionality and production method than it is to describe everything in linear dimensions.

It also provides a communication tool with manufacturing vendors, customers, as well as quality inspectors.

As a result of having a unified vision and language for what they aim to achieve, several departments are able to operate more in parallel..

When done correctly, GD&T can even enable statistical process control (SPC), which reduces product reject rates, assembly failures, and the labour required for quality control, saving businesses a lot of money.

How GD&T Works in automotive designing & manufacturing

All dimensions for a part's properties must be shown on engineering drawings..

A tolerance value with the minimum and maximum allowable limits must be stated.

To appropriately tolerance a product, we need a symbol communicating the design intent.

Moreover, all necessary dimensions of the ultimate product must be stated, with reference dimensions used sparingly.

Parts with unexpected variability and complicated forms necessitate GD&T techniques that go beyond basic plus-minus tolerancing.

The art of tolerancing means to specify just the right variations for all specific design features in order to maximize product approval rate within the limits of the manufacturing processes and depending on the part’s visual and functional purpose.

With more than a few symbols, the datum feature, and the feature control frame, it is feasible to greatly enrich production drawings while while ensuring that engineering fits stay constant across product assemblies.

The designer is able to effectively transmit the design objective while limiting the possibility of it being misconstrued by adhering to the core guidelines of GD&T.

GD&T encourages developers to consider how to best tolerance their components for the chosen manufacturing process, since different manufacturing procedures result in various characteristic deviations.

References

Henzold, G 2006, *Geometrical Dimensioning and Tolerancing for Design, Manufacturing and Inspection, 2nd edn,* Elsevier Ltd, Oxford

Krulikowski, A 1997, *Fundamentals of Geometric Dimensioning and Tolerancing, 2nd edn,* Cengage Learning, Albany

Formlabs. 2021. *The Basics of Geometric Dimensioning and Tolerancing (GD&T)*. [online] Available at: <https://formlabs.com/eu/blog/gdt-geometric-dimensioning-and-tolerancing/> [Accessed 4 June 2021].

Quality-one.com. 2021. *GD&T | Geometric Dimensioning and Tolerancing | Quality-One*. [online] Available at: <https://quality-one.com/gdt/> [Accessed 4 June 2021].

Keyence.com. 2021. *GD&T Fundamentals | KEYENCE America*. [online] Available at: <https://www.keyence.com/ss/products/measure-sys/gd-and-t/> [Accessed 8 June 2021].

Thank You