Product and Manufacturing Information(PMI)

Ravi Krishnan V[1]
Post Graduate Student
Department of Mechanical Engineering
Veermata Jijabai Technological Institute
Mumbai, India
ravi.krishnan30@gmail.com

A.S. Rao[2]
Assistant Professor
Department of Mechanical Engineering
Veermata Jijabai Technological Institute
Mumbai, India
asrao@vjti.org.in

Rahul Rajadhyaksha[3]
Senior Product Manager
Plant 11, 3rd Floor, Pirojshanagar,Vikhroli (West), Mumbai, India
Rahul.Rajadhyaksha@geometricglobal.com

Pankaj Gupta[4]
Senior QA
Plant 11, 3rd Floor, Pirojshanagar, Vikhroli (West), Mumbai, India
Pankaj1.Gupta@geometricglobal.com

Abstract— Use of Product and Manufacturing Information (PMI) in 3D CAD is an emerging field these days. Information regarding Geometric Dimensioning and Tolerancing (GD & T) is of immense importance at every stage from design to manufacture of a product. PMI achieves integration of this GD & T information, annotations, balloons, notes etc. with 3D CAD model. Such integration provides all necessary information at one place. This paper discusses various types of tolerances, complete PMI and its applications.

Keywords- *Computer Aided Design (CAD), Geometric Dimensioning and Tolerancing (GD&T), Model Based Engineering (MBE), Product and Manufacturing Information (PMI)*

*****

I. INTRODUCTION

A finished product is normally defined by 2D drawings. Everyone can easily interpret 2D drawings as they contain standard symbols. However, in some cases, multiple redundant data may be required to define a manufactured part. This may lead to deviations in the final 3D form. Also, manufacturers can subject this data to various interpretations. This can lead to errors in the final product. Moreover, if errors are detected at a later stage manufacturing cost increases. This can also translate into lower quality and productivity.

A simple change in the product can lead to several changes in the 2D documentation associated with it. Maintaining this documentation is time consuming. Therefore, 3D digital data can be used to bring a simple change in the product definition. This led to evolution of a 3D cad model.

However, the 3D cad model was not providing any information required for manufacturing. The manufacturer had to rely on 2D drawing for 'Dimensioning and Tolerancing' purposes. There is large scope for errors in the finished product as the manufacturer had to compare the two models simultaneously.

This resulted in a new concept of providing the complete information in the model itself (2D information on 3D CAD part).The new concept is called as 'Product and Manufacturing Information' abbreviated as PMI. A true manufacturing process requires both the 3D form and 2D information to produce a correct part.

In a 3D CAD model, the digitally stored information can be easily re-used by numerous downstream processes; from the automated creation of 2D drawings to final inspection of the produced part. Understanding and communicating the values of 3D PMI throughout an enterprise (engineering departments, manufacturing floor and at the final supplier base), can help manufacturers realize higher productivity, quality and efficiency gains.

Figure1 shows a part with complete PMI information.

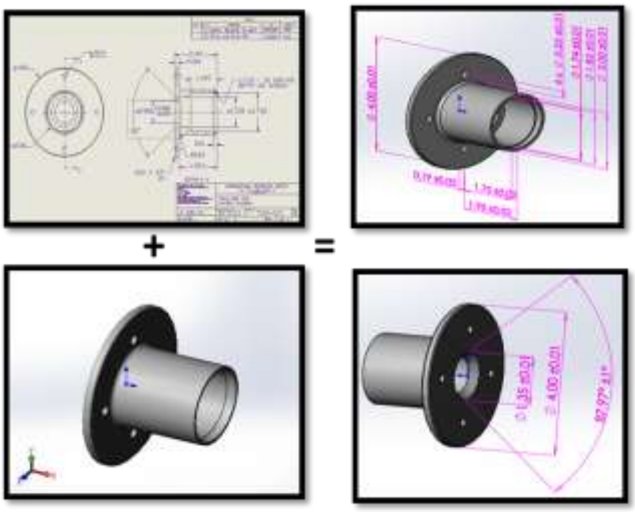

Figure 1.Complete PMI Information on a Part

II. EVOLUTION OF MODEL BASED ENGINEERING (MBE)

The limitations of 2D drawings and 3D CAD Model led to the requirement of new and more powerful engineering software. These softwares will use data models for performing all engineering functions. This necessity gave rise to the concept of 'Model based Engineering'. MBE facilitates the use of models as the data source for all engineering activities throughout the product life cycle.

Product's Model-Based Definition (MBD) is defined as a dataset containing the model's precise 3D geometry and annotations; which specify the product's manufacturing data.

MBD is essential for ensuring the intelligent application of GD&T, since the concepts and symbolic language of GD&T are so complex that few individuals have the time, skill or interest to master them. This lack of mastery leads to improper interpretations, which further leads to most GD&T being useless or even dangerous. MBD promises automating GD&T to part geometry and features. This would enable correct execution of tasks such as tolerance stack-up analysis, manufacturing process management and assembly. Figure 2 shows the evolution in Cad systems from conventional 2D drawings to Model Based Engineering.

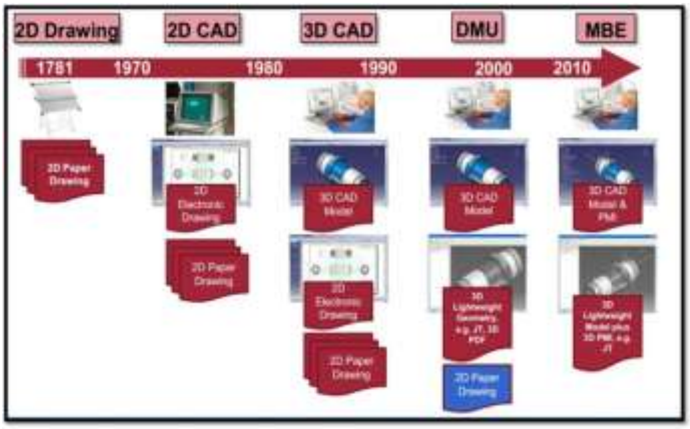

Figure 2.Evolution of Model Based Engineering

III. Product and Manufacturing Information

A product model used in manufacturing is a container of nominal geometry and any additional information required for production and support. This additional information is known as Product Manufacturing Information (PMI). It includes geometric dimensions and tolerances (GD&T), material specifications, component lists, process specifications, and inspection requirements.

It conveys non-geometric attributes in 3D CAD design used in manufacturing product components or subsystems. The data from 3D PMI model drives production and quality processes.

Industry standards for defining PMI include:

1) ASME Y14.41-2003 Digital Product Data Definition Practices and ISO 1101:2004.
2) Geometrical Product Specifications (GPS).

The components of PMI are:

1. Geometric Dimensioning and Tolerancing (GD & T).
2. Datums.
3. Feature Control Frame.
4. Notes.
5. Weld Symbols.
6. Surface Finish Symbols.
7. Balloon.

3.1 Geometric Dimensioning and Tolerancing (GD&T)

Geometric Dimensioning and Tolerancing is a language used on engineering drawings to describe a part accurately. The language consists of a well-defined set of symbols and rules to describe the size, form, orientation, and location tolerances. The need for drawing notes are reduced for describing complex geometry on a component or assembly as GD & T uses standard symbols. It is being used for many years in the automotive, aerospace and the commercial design and manufacturing industries. A reference coordinate system on a component or assembly can be easily defined with the help of GD & T. Geometric Tolerancing specifies the allowable variation for the form and the size of individual features, and the allowable variation in orientation and location between features.

3.1.1 Geometric Dimensioning

In PMI, complete information about the size and shape has to be added on the component or assembly. Adding size information to a drawing is known as dimensioning. PMI follows Baseline dimensioning in which a base line (or datum line) is established for each coordinate direction, and all dimensions are specified with respect to these baselines.

3.1.2 Tolerancing

Tolerance is defined as the permissible limit or limits of variation in a physical dimension. This variation can be in the size and/or the geometry of a component. It is needed because **"No one or thing is perfect"**. Tolerance is also defined as dimensioning for interchangeability as there might be variation in the dimensions during the production of the components at different times. To make the components assemble properly, it requires some standards in tolerance. The two most common standards agencies are:

- American Society Of Mechanical Engineers (ASME)
- International Standards Organization (ISO).

Choosing the correct tolerance for a particular application depends on design intent of the part, method of manufacturing, cost and on individual's experience. Different types of tolerance used in GD&T for size dimensions are enlisted below.

1. Limit Dimensions

Limits are defined as the maximum and minimum size that a part can obtain. The higher limit is placed above the lower limit. When both limits are placed on one line, the lower limit precedes the higher limit. For example, the diameter of a shaft might be specified as follows.

$$\varnothing\ {}^{1.001}_{0.999} \text{ or } \varnothing\ 0.999 - 1.001$$

2. Plus or Minus Tolerances

Plus or minus tolerance defines a basic size and the variation around that basic size. It is further classified as unilateral tolerance and bilateral tolerance.

i. **Unilateral Tolerance**.

In this system, the variation in the dimension of the part is on one side of the basic size. Examples of unilateral tolerance are as follows:

$$25^{+0.02}_{+0.01}\ ,\ 25^{-0.02}_{-0.01}\ ,\ 25^{+0.0}_{-0.01}$$

This system is preferred in interchangeable manufacturing, especially when precision fits are required. It is easy and simple to determine deviations.

ii. Bilateral Tolerance.

In this system, the variation in the dimension of the part is on both the sides of the basic size but may/may not be equally disposing about the basic size. Examples of Bilateral Tolerance are as follows:

$$25^{+0.02}_{-0.01}, 25^{+0.05}_{-0.01}$$

This system is used in mass production when machine setting is done for the basic size.

3. Geometric Tolerance

Different types of tolerances used in GD&T for Geometric Tolerancing are as follows:

1) Form Tolerances

Form tolerances are applicable to individual features or elements of a single feature. Form tolerances are not related to datums. Following are the types of form tolerances:

a) Straightness

Straightness is a form where an element of a surface, or an axis, is a straight line. A straightness tolerance is applicable where the elements to be controlled are represented by a straight line.

b) Flatness

Flatness is the condition of a surface having all elements in one plane. A flatness tolerance specifies a tolerance region defined by two parallel planes within which the surface must lie.

c) Circularity

The roundness of circular parts or features is controlled by circularity. Cones, cylinders and spheres can define these features. The Circularity tolerance specifies where all points of a surface of a circular part must lie. It is a region bounded by two concentric circles where the radius differs by the tolerance value of the concentricity.

Figure 3 gives detailed information of all the type of form tolerances along with their symbols.

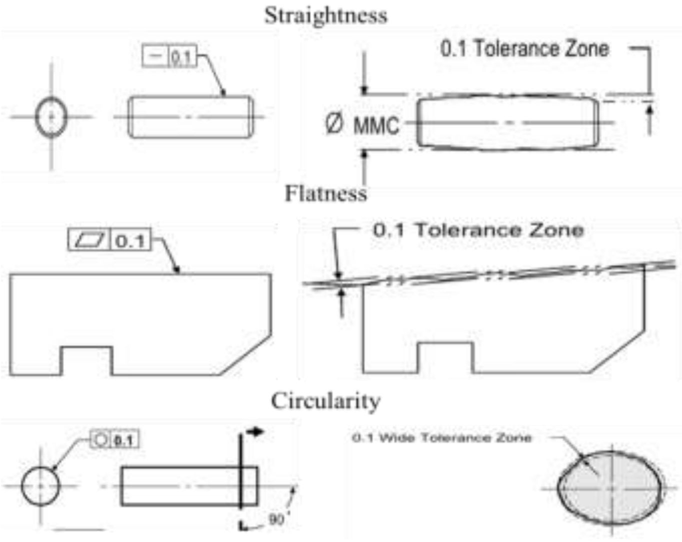

Figure 3.Form Tolerances

2. Profile Tolerance

A Profile is the outline of a part in a given plane. An uniform boundary within where the elements of the surface must lie is defined by profile tolerance. Profile tolerance is applicable in all directions. Hence, it is a three-dimensional tolerance. It can be either unilateral or bilateral. Following are the types of Profile tolerances:

a) Profile of a Line

Profile of a line tolerance governs the profile variation, either unilaterally or bilaterally, along a line element of a feature.

b) Profile of a Surface

Profile of a surface tolerance governs the variation, either unilaterally or bilaterally, on a surface.

Figure 4 gives detailed information of all the type of profile tolerances along with their symbols.

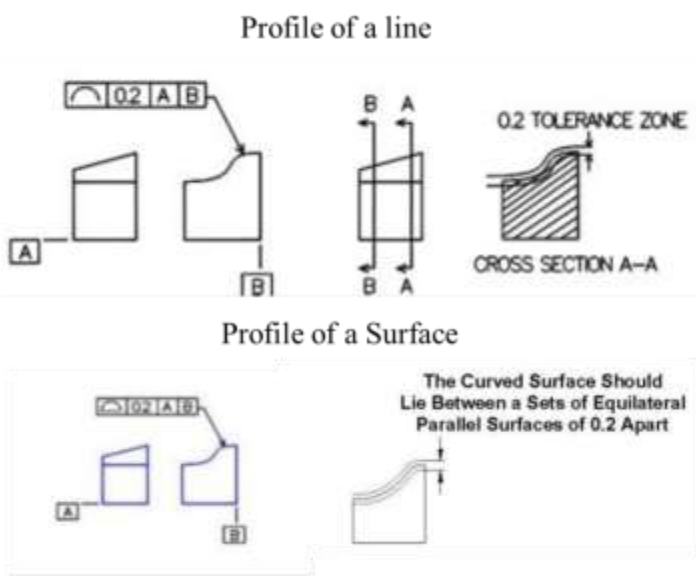

Figure 4.Profile Tolerances

3. Orientation Tolerance

Orientation tolerances control the orientation of features to a datum plane or axis. Following are the types of Orientation tolerances

a) Perpendicularity:

Perpendicularity tolerance controls how much a surface, axis, or plane can deviate from a 90-degree angle. In perpendicularity tolerance, the tolerance region is defined by two parallel planes perpendicular to a datum plane, datum axis, or axis. The surface or the axis of the feature must lie within this region.

b) Angularity:

Angularity tolerance controls how much a surface, axis, or plane can deviate from the angle described in the design specifications. In an angularity tolerance, the tolerance region is defined by two planes at the specified angle, other than 90 degree, from a datum plane or axis. The surface or the axis of the feature must lie within this region.

c) Parallelism:

Parallelism tolerance region is the condition of a surface or center plane equidistant at all points from a datum plane, or an axis. The tolerance specifies the distance between the parallel lines, or surfaces. In parallelism tolerance, the tolerance region is defined by two parallel planes or lines parallel to a datum plane or axis. The surface or the axis of the feature must lie within this region.

Figure 5 gives detailed information of all the type of orientation tolerances along with their symbols.

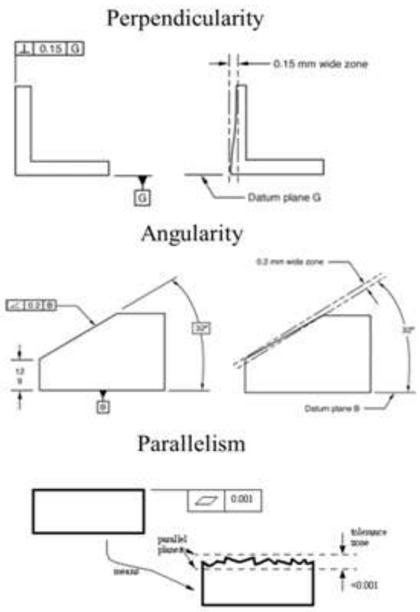

Figure 5.Orientation Tolerances

4. Location Tolerance

Location Tolerances are classified in three types. They are position, concentricity, and symmetry tolerances. The center distance of feature elements are controlled by concentricity and symmetry. The coaxiality of features, the center distance between features, and the location of features is controlled by position tolerance.

a) Symmetry:

The median points of a feature are controlled by symmetry tolerance. It is applied to non-circular features. Symmetry Tolerance governs the variation of the median points between two features from a specified center plane or axis.

b) Position:

Positional tolerance varies along all directions. Hence, it is a three-dimensional geometric tolerance. It governs the deviation in location of a feature from its true position. Features such as holes, keyway can be located from datum planes using positional tolerance. The tolerance defines a region that the axis or center plane of a feature may vary from.

c) Concentricity:

A concentricity tolerance controls a cylindrical tolerance region whose axis coincides with the datum axis. A region within which all cross-sectional axes of the feature being controlled must lie is defined by concentricity. The region is equally oriented about the datum axis for concentricity. In concentricity, the median points of the feature, regardless of its size, must lie within the tolerance zone.

Figure 6 gives detailed information of all the type of location tolerances along with their symbols.

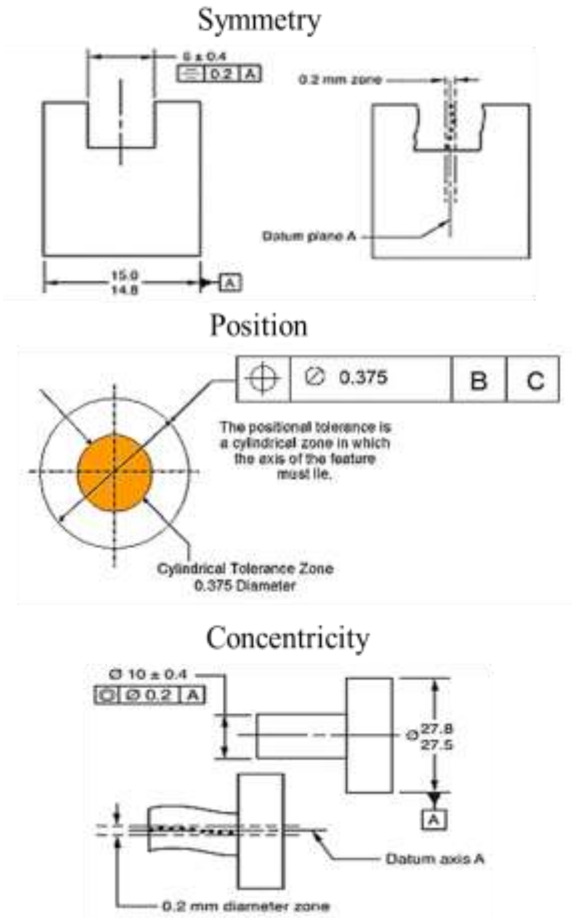

Figure 6.Location Tolerances

5. Runout Tolerances

Runout tolerances are three-dimensional and are applicable only to cylindrical parts, especially for parts that rotate. Following are the types of Runout tolerances:

a) Circular Runout:

Circular Runout can be defined as a two-dimensional geometric tolerance. As cylindrical part rotates circular runout controls the form, orientation, and location across multiple cross sections. It controls the cumulative variation of circularity (roundness) and coaxiality for features constructed around a datum axis. It also controls the circular elements of a surface constructed at an angle not parallel to the datum axis.

b) Total Runout:

In total runout tolerance is controlled along the entire length of, and between, two imaginary cylinders, not just at cross sections. It controls cumulative variations in circularity, coaxiality, straightness, taper, angularity, and profile of a surface. The tolerance region is an annular cylindrical volume of revolution about the center axis of the circle and concentric with the feature surface.

Figure 7 gives detailed information of all the type of runout tolerances along with their symbols.

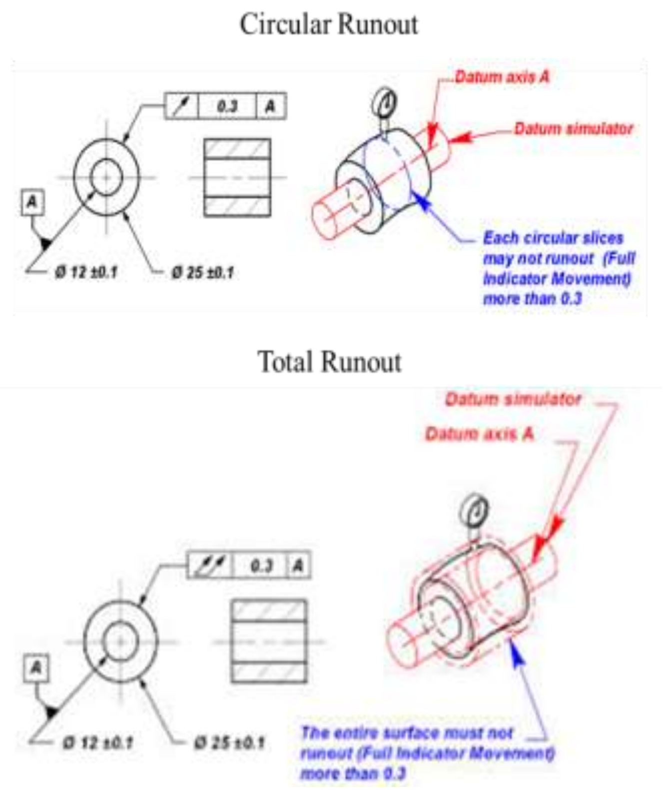

Figure 7. Runout Tolerances

3.2 Datum

A datum is a theoretically a plane, point, or axis from which all dimensional measurement is made in PMI. The variations in the tolerances are also defined with respect to these datums. A datum feature is a physical feature of a part identified by a datum feature symbol and corresponding datum feature triangle. Datum features are selected on the basis of part function and assembly requirements. The datum features are often the features that orient (stabilize) and locate the part in its assembly.Figure 8 shows the representation of a datum on a part.

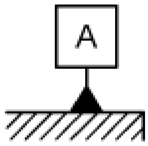

Figure 8.Representation of Datum

3.3 Feature Control Frame

A feature control frame consists of a rectangular frame that is divided into two or more blocks. It is like a basic sentence that can be read from left to right. It defines characteristic type, tolerance shape and value and datum references. The number of compartments in the feature control frame can vary. Figure 9 shows a part with Feature controlled frame which can be interpreted as follows:

The Center of the Hole Feature must be within a 0.05 diameteral tolerance zone at ‘Maximum Material Condition’ relative to datums A and B.

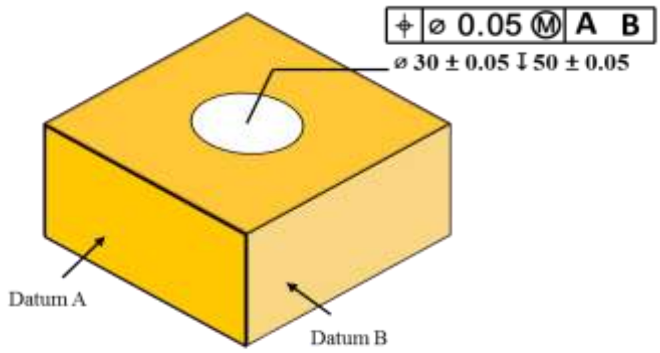

Figure 9.Representation of Feature Controlled Frame

3.4 Notes

Notes are information other than pictorial views and dimensions required for completing a drawing. Any additional information for manufacturing can be put on the part as a note.

3.5 Weld Symbols

Welding Symbol indicates welding joints on engineering drawings. In PMI information will consist of the location, the type of weld and the size and length of the fillet weld.

3.6 Surface Finish Symbols

Surface roughness symbol conveys manufacturing process related information only. Unless written specifically on the symbol, they do not carry the surface texture type (i.e. plated / milled / cold drawn). These symbols are given irrespective of material and its surface condition in PMI.

3.7 Balloon

This phenomenon annotates ‘Balloon’ on a part. Balloons can be used to indicate the part number associated with the ‘Bill of Materials’.

IV. Advantages of PMI

PMI has the following advantages:

1. Time and cost of documenting a part can be reduces.
2. Ensures that design intent is completely captured and associated to the model.
3. Interpreting design intent from 2D information is no longer necessary.
4. Reduces manufacturing errors encountered as a result of manual translations.
5. Rework associated with inaccurate or incomplete manufacturing information is reduced.
6. Increases productivity and quality by documenting the information once and reusing it everywhere.

V. Applications of PMI

1. Computer Aided Manufacturing (CAM) software can directly use a 3D model with PMI and can directly generate tool path considering all necessary factors. This is supposed to reduce human intervention in CAM process resulting into timesaving and error free output.
2. PMI output can be used as a direct input for inspection process. For automatic inspection system it can result into efficient execution and for manual inspection it can reduce human confusion.

VI. Conclusions

1) PMI is a very promising technology.
2) It can enhance dialogue between design and other departments of an organization.

VII. References

[1] Alex Krulikowski. "Fundamentals of Dimensioning and Tolerancing",1997.

[2] Daniel E. Puncochar. "Interpretation of Geometric Dimensioning and Tolerancing" .

[3] James D.Meadows."Geometric Dimensioning and Tolerancing Workbook and Answerbook".

[4] Gene R. Cogorno." Geometric Dimensioning and Tolerancing".