Solving Issues Using Improved Design Techniques

Picture Courtesy: www.solidworks.com

Quality issues, re-work, warranty problems and consistency in manufacturing are challenges faced by the industry irrespective of whether it is a limited volume or mass production. This affects profitability, sustainability and above all customer trust. In order to become world-class organizations in manufacturing, Indian companies need to cut down on rejections, improve product consistency and re-invest in ways and means to augment technology as a part of the continuous improvement processes.

Importance of drawings

Every department in a company, associated with product development, namely, design, manufacturing, quality, purchase, service, vendor development, needs to access and interpret drawings. An error in the drawing affects the organization as a whole, in terms of time, resources and cost. It is important that

Poor quality in assembly build and performance significantly impacts profitability. Achieving acceptable quality often depends on the drawings that define design specifications. GD&T drawings developed according to ASME Y14.5/ ISO standards and ensured for correctness and completeness is the first step to better quality. Assigning least cost tolerances, and predicting assembly build and performance is the next step towards developing a foolproof quality regime. This is done through Tolerance Stack Up Analysis.

BY **NATARAJAN R**

the drawings are correct and complete in all aspects of dimensioning and tolerancing, leaving no ambiguity in interpretation by cross-functional teams.

Problems faced by manufacturing and inspection are often a result of drawings that are incomplete – in terms of dimensions and, more importantly tolerances. In spite of investing in expensive machines to maintain critical tolerances and having top-of-the-line inspection systems, if the tolerances are not addressing assembly build and performance requirements, rejections and unsatisfactory customer experiences are inevitable.

It is important that the design engineers need to allocate functional tolerances to ensure assembly build and consistent performance. Needless to say, what goes on the drawing affects the profitability of the business directly.

Tolerance allocation—cost implications

Normally, tolerances are allocated at part level based on past experience, guideline values provided by handbooks and, sometimes, 'gut feel.' Rarely tolerances are provided based on assembly implications, build, functional requirements, process capability (Cp, Cpk) and above all cost. A quality problem will, in most cases, result in overall tightening of tolerances, more inspection dimensions and increased checks at more stages and frequencies. A direct result of this is increased cost, lesser profitability and more rejections. It is important, therefore, to allocate tolerances according to fit, form and functional requirements. This is possible by incorporating Geometric Dimensioning and Tolerancing (GD&T) and Tolerance Stack Up Analysis as a part of the design process.

GD&T: assurances and benefits

Drawing standards have gained importance with the evolution of GD&T practices. ANSI Y14.5 (1994) has gained worldwide acceptance and is a forerunner to the ISO standard. Many manufacturing companies are adopting Y14.5 on account of savings in cost, time and effort.

Let us take the following example to assess the advantages.

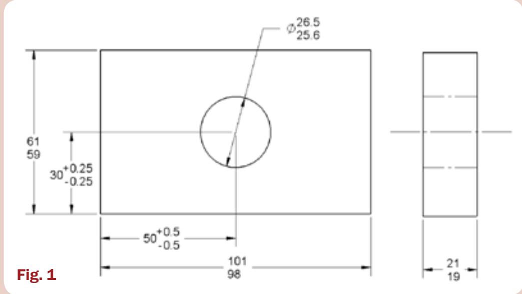

Fig. 1

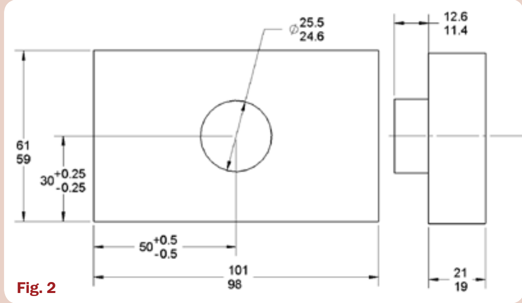

Fig. 2

Fig. 1: Conventional drawing approach

Fig. 2: Conventional drawing approach—Mating part

The drawing shown in Fig. 1, draws more questions than answers provided. Here are a few:

1. What is to be located and measured, hole axis from edge or edge from hole axis?
2. How is the part to be set up for manufacture and inspection?
3. Why is the tolerance zone for the location of hole axis rectangular, when the hole shape it is projecting is circular?
4. What about the mating part? How to ensure assembly all the time?

GD&T approach uses a four-step process keeping assembly and function in perspective.

1. Identification of origins of reference and directions for location and measurement (Datums)
2. Assignment of nominal (ideal) dimensions for part features
3. Definition of tolerance zones for features
4. Dynamic interaction of tolerance zones to maximize tolerances

Though the size tolerance is lesser in the mating pin, as shown in Fig. 2, as compared to the internal feature of size in Fig. 1, there is no assurance that all

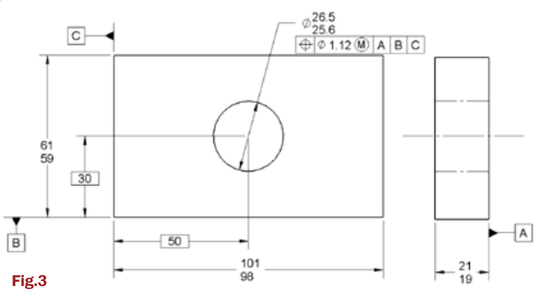

Fig.3

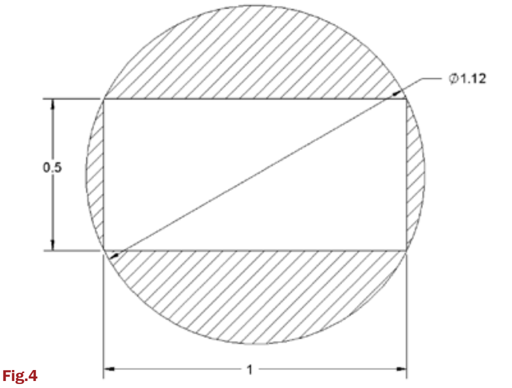

Fig.4

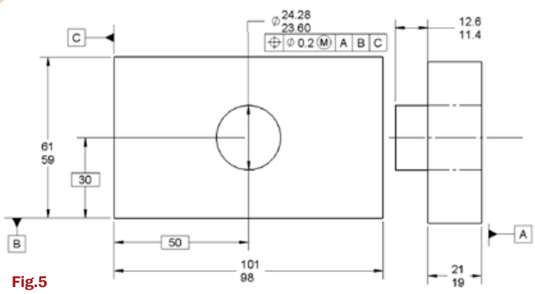

Fig.5

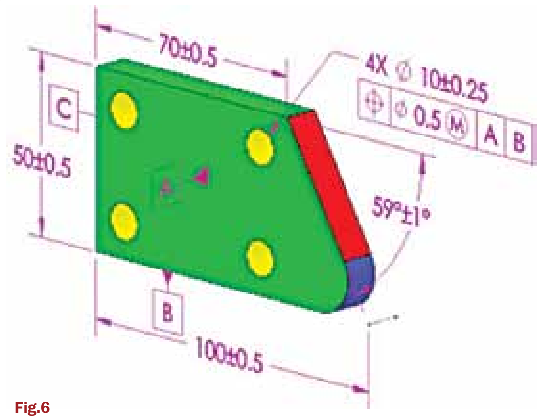

Fig.6

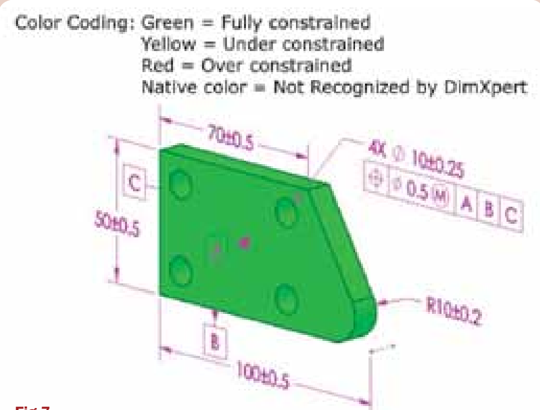

Fig.7

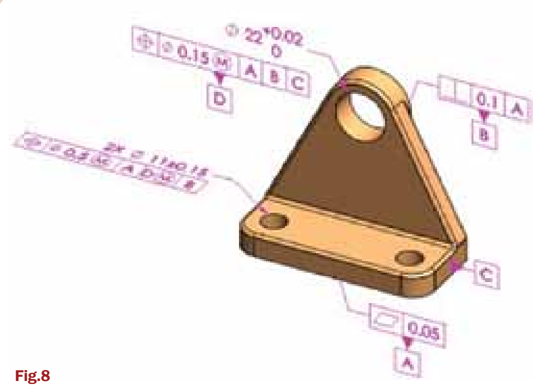

Fig.8

Fig. 3: Drawing approach based on GD&T

Fig. 4: Area analysis of GD&T and plus/minus tolerance zones

Fig. 5: Mating part GD&T – Assuring assembly

Fig. 6: Colour coded tolerance conflict evaluation

Fig. 7: Colour coded tolerance conflict resolution

Fig. 8: Leveraging 3D annotations with DimXpert in SolidWorks

mating parts would assemble as desired. The feature of size (Level 1 control—diameter) as well as the location (Level 4 control—position) together influence assembly.

Converting of the stated drawings to GD&T standard ensures assembly and reflects real-world conditions, while maximizing tolerances.

As shown in Fig. 4, if we look at the area of the proposed tolerance zone, per GD&T, vis-a-vis conventional approach, increase in tolerance zone is evident. Bonus tolerance is available as the feature of size departs from MMC towards LMC due to the modifier (M). If the pin confines to the virtual boundary of Ø 24.48, then assembly is assured. One of the mating part dimensions that ensures assembly is shown in Fig. 5.

The values used in tolerances of size and locations in the mating parts are only an example, to demonstrate the efficacy and versatility of GD&T in terms of the following benefits:

- Apportioning tolerances according to process capability
- Assembly always happens
- More tolerance than conventional method at lesser cost

It is important to note that the drawings need to be correct and complete in tolerance definition to avoid ambiguity and mis-interpretation (that more often than necessary result in re-work and cost burden). Contemporary design software such as SolidWorks, using DimXpert, ensure both correctness and completeness of drawings are assured according to ANSI Y 14.5 standard, as shown in Fig. 6.

Predicting assembly build using Tolerance Stacks

GD&T is incomplete without incorporating Tolerance Stack studies as a part of tolerance definition and allocation at part level. Assembly build and performance criteria are addressed by Tolerance Stack Up Analysis. Let us take the example of an electric motor as shown in Fig. 9. Tolerance variations at part level will affect assembly and performance and efficiency.

Tolerance Stack up Analysis can be performed in 1-D, 2-D and full 3-D to study the dynamic interactions of the tolerance zones.

Section view of a motor with dimensional layout is provided in Fig.11. As first step, we need to ascertain what the gap variation would be for the allocated dimensions at the part level. This is called Roll-Up Analysis. To proceed further, it is important to identify the 1-D chain of dimensions affecting the assembly build objective. The dimensions are shown in Fig. 12.

Worst case tolerance stack up analysis

Description	Variable Name	Mean Dimension	Sensitivity	Fixed/ Variable	±Equal Bilateral Tolerance
Screw Thread Length	A	37.5	-1	Fixed	0.3
Washer Thickness	B	5	1	Fixed	0.05
Length of Inner Bearing Cap	C	7.5	1	Variable	0.1
Bearing Width (Thickness)	D	35	1	Fixed	0.3
Spacer Thickness	E	10	1	Variable	0.1
Rotor Length	F	115	1	Variable	0.25
Spacer Thickness	G	10	1	Variable	0.1
Bearing Width (Thickness)	H	35	1	Fixed	0.3
Pulley Casting Thickness	I	32.5	1	Variable	0.3
Shaft Length	J	227.5	-1	Variable	0.5
Tapped Hole Depth on Shaft	K	20	1	Variable	0.2
Gap		5			2.5

Stack Analysis Table can be generated, as above, to calculate Stack up by Worst Case Method (WC). The Tolerances Stack Up calculation involves definition of Node-Tree based on Dimensions influencing assembly build, as shown in Fig. 13.

Sub-nodes are used for parts having multiple feature dimensions influencing assembly build. Roll-Up Analysis, per SigmundWorks, predicts a gap of 5 ±2.5mm. These are called the Control Limits, based on the part dimensions provided. Specification Limits, as required by the assembly build, are 5±1mm. Statistical report of PPM with processes following normal distribution is given as in Fig. 14. Sensitivity analysis of the dimensional tolerances influencing assembly build is reported as shown in Fig. 15.

It is important to note that the Sensitivity Analysis gives a deep insight into the functional dimensions that affect assembly build. This is beneficial in two ways, namely:

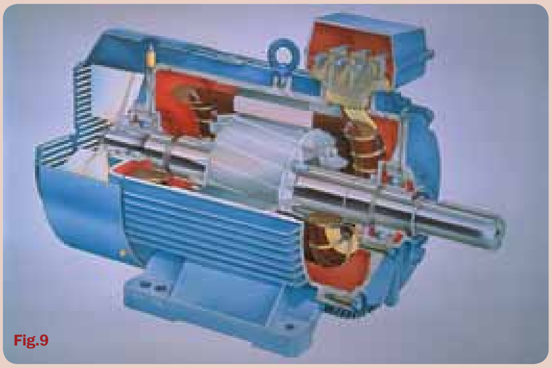

Fig.9

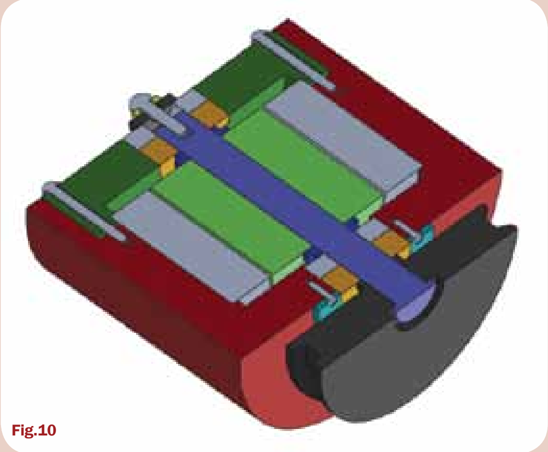

Fig.10

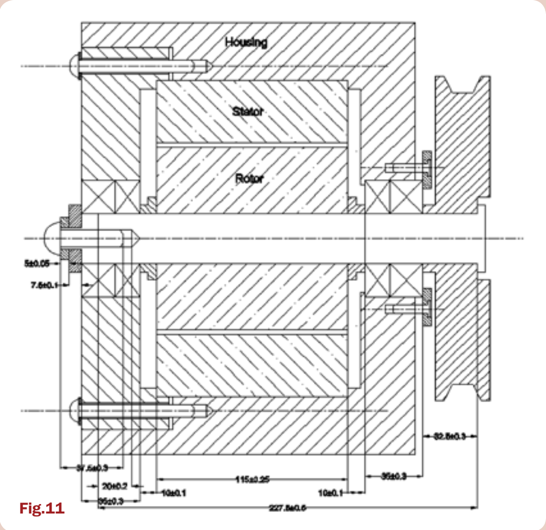

Fig.11

Fig. 9: Cut section view of an electric motor assembly

Fig. 10: SolidWorks CAD model of motor assembly

Fig. 11: Motor section drawing with linear dimensional layout

Roll-down Calculation

Description	Variable Name	Mean Dimension	Sensitivity	Fixed/ Variable	±Equal Bilateral Tolerance	Roll-Down Tolerances to meet Build Objective
Screw Thread Length	A	37.5	-1	Fixed	0.3	0.3
Washer Thickness	B	5	1	Fixed	0.05	0.05
Length of Inner Bearing Cap	C	7.5	1	Variable	0.1	0.008
Bearing Width (Thickness)	D	35	1	Fixed	0.3	0.3
Spacer Thickness	E	10	1	Variable	0.1	0.008
Rotor Length	F	115	1	Variable	0.25	0.008
Spacer Thickness	G	10	1	Variable	0.1	0.008
Bearing Width (Thickness)	H	35	1	Fixed	0.3	0.3
Pulley Casting Thickness	I	32.5	1	Variable	0.3	0.008
Shaft Length	J	227.5	-1	Variable	0.5	0.004
Tapped Hole Depth on Shaft	K	20	1	Variable	0.2	0.004
Gap		5			2.5	0.998

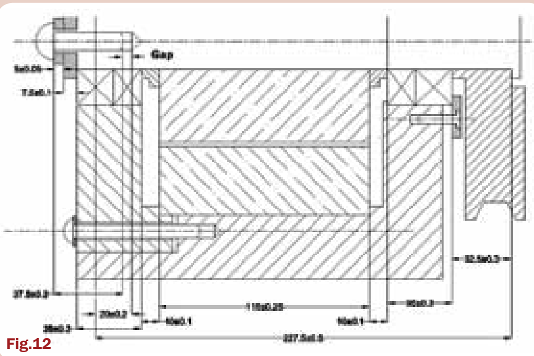

Fig.12

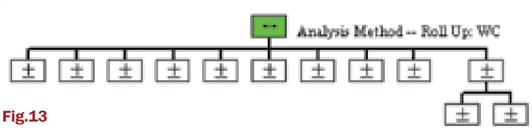

Fig.13

Fig.14

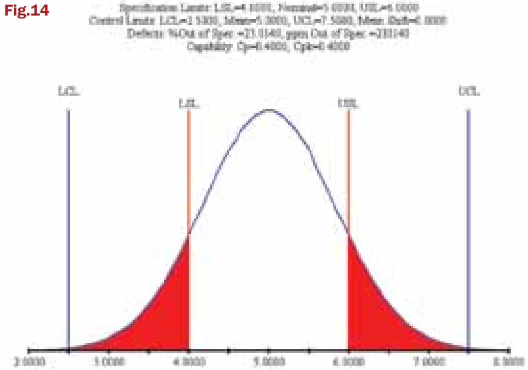

Fig.15

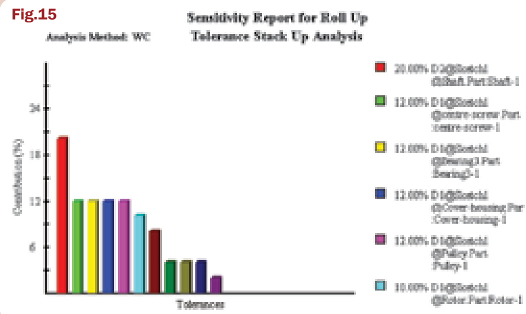

Fig. 12: Gap definition with the influencing dimension layout

Fig. 13: Node-Tree definition using SigmundWorks in Solidworks

Fig. 14: Roll-up statistical report using SigmundWorks

Fig. 15: Sensitivity report for contributing dimensions using SigmundWorks

- Objective choice for the user to allocate tolerances based on least cost and process capability while maximizing tolerances
- Identification of processes in manufacturing that need to be inspected when the process goes out of control to ensure process correction

In the current study, we need to arrive at part dimensional tolerances that would achieve 5 ± 1mm, while working within the constraints that bearings, washers and screws are standard catalogue items having specified limits of sizes.

Arriving at individual dimensional tolerances based on specified assembly build objective is called Roll-Down Analysis. Specifying the stated limits of sizes as fixed, per table, a Roll-Down Calculation is performed by specifying build objective as 5 ± 1mm.

Remarks

In this example, allocated tolerances are not meeting the build objective. This would result in over 23 per cent rejections that will result in increased cost and re-work. While imposing constraints on tolerances of bought-out items, the tolerances are equally distributed between contributing dimensions in the absence of process capability and cost constraints. This example is just to demonstrate the efficacy of Tolerance Analysis in achieving build objectives. The analysis is for Worst-Case, wherein the process has liberty to approach extreme limits of sizes usually resulting in stringent tolerance specifications. ■

Natarajan R is the Director of EGS Computers India.