中國機械工程學刊第四十五卷第三期第 219~228 頁(民國一百一十三年)
Journal of the Chinese Society of Mechanical Engineers, Vol.45, No.3, pp219~228 (2024)

A Method of Reducing Tolerance Stack-Up For Parts Assembling of Molds

Yuo-Tern Tsai* and Chun-Sheng Chen**

Keywords: tolerance, positioning, mold.

ABSTRACT

Allocating appropriate tolerance to the components to meet the functional requirements of products is an important task in the design phase. This article introduces the methods of tolerance analysis including tolerance marking for improving the assembly precision for injection molds. The influences of modification symbols of geometric tolerance are reported to support tolerance designing. The dimensional chains of the mold are constructed for performing tolerance analysis and design improvements. The statistical methods of tolerance analysis are used to evaluate the unqualified rates of the mold under different tolerance designs. The unqualified rates are estimated based on the precision requirements and the natural tolerances of the manufacturing capability. A method of assembly positioning is proposed for the mold to reduce the stacked tolerances as well as improving its assembly precisions. The studied results show that the total tolerances of the mold can be reduced by the modified designs. The unqualified rates of the mold can be improved from 5.76% to 3.38% for the sane precision requirements and the tolerance designs. The study offers a systematic method of tolerance designing and analysis to ensure the designed tolerances satisfying the functional requirements.

INTRODUCTION

Following industrial technological developments, product designing is becoming more and more diverse and refined. Integrating engineering analysis with probabilistic evaluations to perform reliability design is an efficient approach to ensure the designed quality (Tsai et al., 2013). For plastic products, molds are commonly adopted to fulfill mass manufacturing. Mold production possesses the properties of high repeatability and low cost in manufacturing. It is the foundation of forming industry and has the technical specificity in mechanical manufacturing. Mold technologies can be regarded as the base of precision manufactures and the development of the machinery industry. Mold designing plays a critical role in the precision machining and has a considerable impact on the quality of the produced parts. The developments of mold manufacturing are requested not only the precision but also the finished time as short as possible to promote the market competition (Tsai and Chen, 2022).

It is impossible to obtain a product with the exact dimensions while processing due to the limits of machine precision. A certain degree of dimensional variations is always existing in mechanical processing. The sources of the dimensional variations usually are brought out by processing tolerances, such as (1) the variations of the processing tools and the fixtures including tool wear, etc.; (2) Material unevenness, internal stress releases, and surface size variations; (3) improper operational processed and assembly; (4) stiffness and repeatability of processing machines such as the variations of operating parameters (feed, speed, etc.); (5) the influence of ambient temperatures and humidity, etc. If a product is designed with better precisions, the higher manufacturing costs will be incurred accordingly. The considered factors in tolerance designing usually are the trade-off between the precision requirements and the manufacturing costs.

Tolerance representing dimensional variations of a design is either the worst conditions or a range of statistical measurements. Tolerance designing is to set proper tolerances for the parts so that the parts can be processed and then assembled as a whole to satisfy the functional requirements. For an assembly, tolerance and fit are the major contents in design considerations because they describe the specifications of the dimensional variations and the states of the parts assembled (Bjorke,1989). Tolerance in design can be divided into both, tolerance analysis and tolerance allocation. Tolerance analysis is to calculate the total tolerances of an assembly according to the tolerance specifications of the parts. Tolerance allocation is to distribute the tolerances set in the functional requirements into the related parts. Tolerance designing must satisfy the assembly yields, meanwhile, considering the manufacturing costs. If the tolerances of the assembly

Paper Received September, 2023. Revised January, 2024. Accepted February, 2024. Author for Correspondence: Yuo-Tern Tsai
Email: yttsai@mail.hdut.edu.tw

* *Professor, Department of Mechanical Engineering, Hungkuo Delin University of Technology, 236, Taiwan, R.O.C.*

** *General manager, YCI Industrial Co., Ltd.*

exceed the specifications, the tolerances must be redistributed through tolerance allocation to meet the requirements and achieve a balance between the functionalities and the manufacturing costs (Chase, 1999).

The general methods of tolerance designing and analysis can refer to the book (Fischer, 2011). The tolerance diagram is an effective diagnostic tool because it can provide tolerance analysis in the early stages of product development. Tsai & Wu (2013) studied the methods of optimizing the tolerance configuration of machine tool spindles and designed a software for tolerance analysis to keep the assembly tolerance at a high-quality level. Huang et al. (2018) introduced a system that combines process tolerance estimation and error trend monitoring. It allows mechanical and process engineers to establish the process flow in a simple way, and automatically generates dimensional chains and processing tolerance suggestions as the inspection benchmark for finished products. Bacharoudis et al. (2020) formulated the tolerance allocation scheme as a reliability-based optimization problem and developed a probabilistic framework to allocate tolerances to the various features of a product.

Tolerance stack-up analysis is to calculate the accumulated variations across a set of dimensions. It must select the schemes of the dimensions and the tolerances of the related parts and observe the tolerance variation of the assembly. Cao et al. (2018) briefly presented eight of the most widely used models for tolerance analysis. A comparison was proposed to show each method's advantages and disadvantages, similarities, and differences. Conventional methods in tolerance stack-up analysis usually involve many rules and conditions. Sahani et al. (2014) suggested a method to solve the problems of tolerance stack-up involving geometric characteristics. Otsuka & Nagata (2015) proposed a numerical method using Monte Carlo simulation and statistics rule to solve the problems of tolerance stack-up in statistical indices. Tsai et al. (2015) investigated the effect of grouping for components with uniform and normal distributions by the developed "grouped random assembly" method. Focusing on tolerance designing of molds, Tsai et al. (2023) proposed a block assembly method for the mold core which is consisted of a lot of the same parts to reduce the stacked tolerances in assembly.

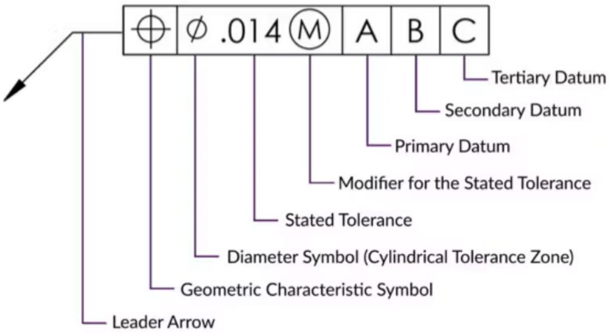
The proposed method for reducing the stacked tolerances in assembly by using assembly positioning method in this paper is different with the method of Tsai et al. (2023) which is by using block assembling. In this paper, a method of assembly positioning is proposed for redefining the dimensional chains so as to the total tolerances of an assembly can be reduced. The assembly of an injection mold is used as an example to depict the method and to identify the improvements in tolerance stack-up. The markings of geometric tolerance and the influences of the tolerance modifiers to the tolerances are introduced for giving proper tolerances in engineering designs. The methods of tolerance analysis and tolerance allocation are also reported to carry out tolerance calculations for the assemblies. The unqualified rates of the mold under various tolerance designs are investigated based on statistical tolerance analysis. The studied results show that the unqualified rates of the mold at the same tolerance designs can be improved from 6.37% to 3.86% by the proposed method. It is useful in reducing tolerance stack-up and developing tolerance-based reliability design for the assemblies.

TOLERANCE MARKING

The notations of traditional tolerance marking majorly describe an allowable variation of the lengths or angles of the parts at a certain direction. The common cognition of tolerance is that if the dimensions of a design are between the maximum and minimum values, it will meet the functional requirements and can be assembled or used normally. The general used methods in tolerance marking have three, dimensional limits, one-way tolerance and two-way tolerance. The dimensions not tolerance marked indicate the general tolerances which are often expressed with annotations in engineering drawings. This expression not only simplifies the complexity of tolerance marking but also letting the manufacturers and inspectors easily to grasp the allowable variations of the key dimensions (ISO 286-1, 2010).

The requirements of accuracy of mechanical products become stricter following the advancement of industry. The traditional tolerance marking methods have loose constraints on the shape of parts, and may cause disputions in some dimensional interpretations on many occasions. To improve these faults, the geometric tolerance is adopted to describe the appearance characteristics of the parts in many designs. It is a more effective tool in expressing the tolerances for industrial products. ANSI standard (ASME Y14.5, 2009) regulates the annotation of dimensions and tolerances, and the declaration of geometric tolerances mainly depends on the annotations of feature control frames and reference frames. The complete feature control frame is shown in **Figure 1**.

The frame body is a long strip of squares and is divided into three areas separated by solid lines. From left to right are geometric tolerance symbols, tolerance ranges and reference datum. There are 14 kinds of general geometric tolerance symbols. In addition to the declaration of the tolerance range, it is necessary to specify the given tolerance amount, and appropriate modifier symbols can be added before and after the tolerance amount depending on the situation. Before the tolerance amount, the possible additional symbol is ∅, which designates the tolerance area as a circle or a cylinder, and the modifier symbol after the tolerance amount mainly describes the retention status of the material referenced when setting the tolerance amount. The rightmost area of the feature control frame is the reference datum. The sequence from left to right represents the reference order. The number of reference datum can range from 0 to 3 according to the nature of the geometric tolerance, and each datum is also separated by a solid line.



SYMBOL	CHARACTERISTICS	CATEGORY
—	Straightness	Form
⏥	Flatness	
○	Circulatity	
⌭	Cylindricity	
⌒	Profile of a Line	Profile
⌓	Profile of Surface	
∠	Angularity	Orientation
⊥	Perpendicularity	
//	Parallelism	
⌖	Position	Location
◎	Concentricity	
⌯	Symmetry	
↗	Circular Runout	Runout
⌰	Total Runout	

Figure 1. Feature control frame

One of the most important features of geometric tolerances is the establishment of datums. In theory, the datum is a point, a straight line, a plane or cylinder, or the axis of a shape. These ideal geometric elements are the reference of the geometric shape of the machine part and the basis of tolerance measurements. these geometric features such as end faces, hole faces, and keyways can be referred as reference datum called datum features. To clearly define the direction and position of a machine part in three-dimensional space, three datum planes are usually required. When the machine part has symmetry, the required datum planes can be reduced. The order in which the datum planes are set has a great influence on the constraints on the shape of the machine part. To ensure that design and manufacturing adhere to a common measurement method, the sequence and precise specifications for establishing benchmarks must be marked on the design drawing. The description of the constraints that all machine parts should have relative to the datum is the declaration of geometric tolerances.

The mark of the datum is a filled or hollow triangle on the projection line or extension line of the shape, and then the vertices of the triangle are connected to the datum code of the box in the vertical direction. The code is conventionally written in uppercase English letters, and the priority of benchmarks is related to the declaration method and has nothing to do with the alphabetical order. **Figure 2** provides only a pictorial representation of the datum notations; the sequence of references must be supplemented by additional notations.

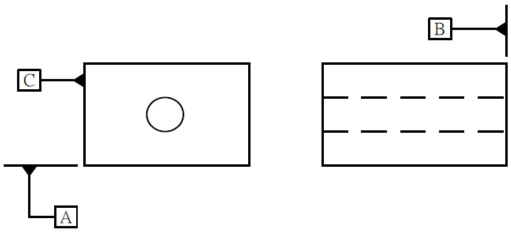


Figure 2. Marks of datum

When the reference shape contains a change in size or the feature itself involves a change in size, it may be necessary to specify the material retention of the reference shape when marking geometric tolerances. According to the customary marking method, three kinds of material retention states can be specified, which are the Maximum Material Condition (MMC, symbol as Ⓜ), the Least Material Condition (LMC, symbol as Ⓛ), and regardless of size, referred to as RFS.

Generally, the most commonly used notation is the condition with the most material remaining. In this case, when the sample size of individual parts has not reached the allowable limit value, the geometric tolerance zone is allowed to be moderately relaxed, and the size of the relaxation is exactly equal to the difference between the sample size and the limit size. The shaft in **Figure 3** is constrained by the diameter and straightness tolerance, and the straightness constraint is affected by the size of the shaft itself.

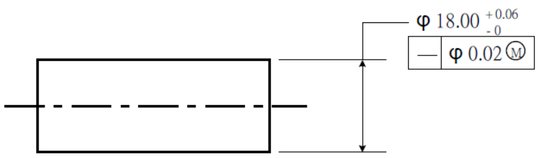


Figure 3. Tolerance modifiers

The modifier symbol Ⓜ after the straightness tolerance means that the straightness requirement of 0.02 mm only limits the maximum size of the shaft (18.06 mm) applied. When the actual size of the shaft is less than 18.06 mm, the allowable variation range of the straightness of the shaft will be greater than 0.02 mm. For example, the allowable straightness tolerance for the shaft 18.05 mm can be enlarged to 0.03 mm; the shaft 18.04 mm can allow the straightness tolerance to 0.04 mm, and so on at the smallest shaft (18.00), the straightness tolerance can be enlarged to 0.08. The increased tolerance called as bonus tolerance is 0.06 mm.

The object of geometric tolerance control is the center position of the feature hole. There is a modification symbol Ⓜ in **Figure 4** next to the label of the reference datum A. The label in the figure indicates that the position of the center axis of the hole is allowed when the datum A is in the maximum physical state (44.8 mm). The tolerance range is a cylinder with a diameter of 0.2 mm. When the position of the actual datum is shifted up to a position 45.0 mm from the bottom, the modified tolerance of the bore axis will be enlarged to a cylinder with a diameter of 0.4 mm. If the datum A is at the highest position (45.5 mm), the modified tolerance of the hole axis will be enlarged to a cylinder of 0.9 mm. The bonus tolerance is 0.7 mm.

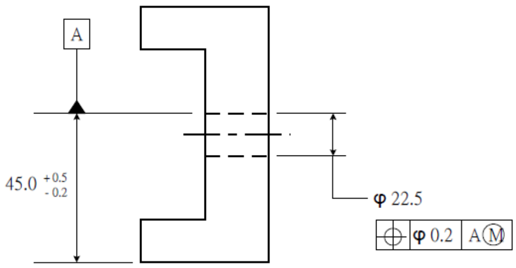


Figure 4. Reference Datum with modifiers

The second material retention situation is similar to the previous description. The symbol Ⓛ represents that the range limit of the geometric tolerance is set according to the minimum physical condition of the datum shape or the reference shape. When the actual size of the part deviates from the minimum physical condition, the range of the geometric tolerance will be relatively enlarged. This type of modifier is usually used to ensure that the certain features (such as wall thickness) have dimensions greater than a certain lower limit. As for the third case, the range of geometric tolerance remains unchanged regardless of the size change of the reference feature. In the new standard, all tolerances shall be treated in this way if no modifier is indicated.

TOLERANCE ANALYSIS

Tolerance analysis can be classified into two types according to the differences in calculation. Tolerance allocation is to allocate the assembly tolerance to the related components so that the components can meet the functional needs and be manufactured accordingly. Several frequently used methods in tolerance allocation are reported here.

Dimensional Chain

The first step of performing tolerance calculation is to construct the dimensional chain. A dimensional chain is the process of an assembly or the mutually connected dimensions of the components to form a closed-dimension group. Generally, when fulfilling mechanical design, it is necessary to perform precise analysis and calculation of the components, meanwhile, reasonably determining the geometric and dimensional tolerances for fitting. Ensuring the components can be assembled correctly, and meet the predetermined requirements. In the assembled processes, it is often encountered a group of dimensions with internal relations indicates the functional characteristics. To fulfill tolerance analysis, the dimensional chain is first created according to the assembled relationship (Bjorke, 1989). The dimensional chain of an assembly can refer to **Figure 5.** The nominal dimension of the gap can be calculated by

$$A_0 = \sum v_i A_i = -A_1 - A_2 + A_3 - A_4 - A_5 \qquad (1)$$

where v_i is the direction of the dimensions (either -1 or 1).

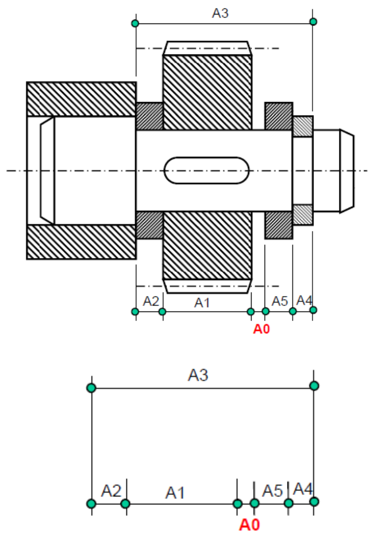


Figure 5. Dimensional chain

Tolerance Calculation

The most commonly used methods in tolerance calculation have two, the Worst-Case Tolerance (WCT) and statistical tolerance (Root Squared Sum, RSS). The WCT is the traditional method of tolerance analysis. The variables each are set to their tolerance limits so that the measurements of an assembly are either maximum or minimum. The statistical tolerance (RSS) utilizes the theories of statistics for relaxing the part tolerances and not sacrificing the quality. The variation of each part is modeled as a probabilistic distribution and these distributions are used to evaluate the variations of the assembled measurements (Fischer, 2011).

The Worst-Case (WC) method sets the variables to their respective upper-lower limits and doesn't consider the individual distribution of the variables. The WC method can obtain the maximum variations of the measurements regardless of the part's variations. The WC method should be used in tolerance designing to obtain 100 percent of the parts assembled and functioning properly. The WCT for an assembly is defined as

$$T_{a,WC} = \sum |v_i| T_{i,WC} = \sum T_{i,WC} \tag{2}$$

where $T_{i,WC}$ are the tolerances of the parts. The major fault in the WC method is that the tolerances of the individual parts must be set very tight. This condition would result in high manufacturing and inspection costs and/or large rates of scrap. The WCT is commonly used in important interfaces and/or spare parts.

The statistical method (Root Squared Sum, RSS) estimates the tolerances using the variations of a distribution, not the specified upper-lower limits. This method offers a measurement in increasing the design flexibility which allows the quality designed with different levels, less than 100 percent. The statistical method permits the non-perfect consistency (less than 100% acceptance) so that the tolerances of the related parts can be enlarged to result in a reduction of the manufacturing costs due to the parts being easily processed. The statistical method usually is applied in mass production because the manufacturing costs are more economical.

The statistical tolerance is always associated with the process capability C_{pk}. While considering the influence of process capability, the part tolerances can be setted according to the tolerance specifications and the natural tolerances of manufacturing (3σ). The part tolerances are modified as

$$T_{i,RSS} = \frac{\mathrm{T}_i}{\mathrm{C}_{pk}} \tag{3}$$

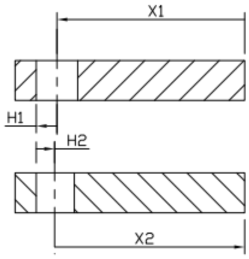
where T_i is the designed tlerance. The statistical tolerance of an assembly can be expressed as

$$T_{a,RSS} = \sqrt{\sum (T_{i,RSS})^2} \tag{4}$$

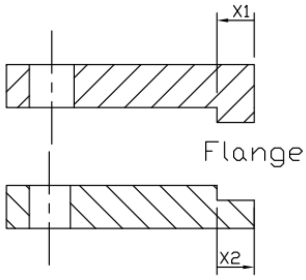
TOLERANCE IMPROVING

Tolerance stacking is commonly to exist in the components which are consisted of many parts. The methods of reducing tolerance stack-up are either to decrease the number of the parts or to increase the precision of the parts as well as decreasing the tolerance ranges. The former must change the structural design so that the component's functions can be accomplished using the less parts as well as decreasing the variables of the dimensional chains. The later always involve an increase of manufacturing costs because the parts must be processed using more expensive machines to meet the tolerance needs. Which one is better depended on the trade-off of the costs and the functional requirements.

A feasible method to reduce tolerance stack-up is by positioning designs. An example of two plates assembled with blots (no display in the figure) is shown in **Figure 6.** The needs of the design are to align the two plates no deviation at the right base sides. The tolerance variables of the bolt holes have two, the position deviations (X) and the hole tolerances (H). In **Figure 6 (a)**, the bolt holes have two functions, fixing and positioning, the total tolerances of the assembly would be $T_a = T_{X1}+T_{H1}+T_{H2}+T_{X2}$ according to the dimensional chain. In **Figure 6 (b)**, the bolt holes just for fixing, the assembly positioning is accomplished by the convex-concave flanges (X1 and X2) at the right side. The total tolerances of the assembly would be $T_b=T_{X1}+T_{X2}$.



(a) The flat design,



(b) The flange design

Figure 6. Two plate assembly

The number of the variables of the dimensional chains are changed from 4 to 2 while the structures are modified. If the machining precisions of the features are the same (the same tolerances), the tolerance stack-up of the flange design would be obviously less than that of the flat design. The phenomenon explains that the total tolerances can be improved by adding the positioning designs.

Table 1. The dimensional tolerances (mm)

Feature Size:	1–20 mm	21–100 mm
ABS/PC	0.05	0.1
HDPE	0.075	0.11
PA	0.03	0.13
PC	0.03	0.1
PMMA	0.05	0.07
POM	0.03	0.13
PP	0.075	0.11
SAN	0.05	0.1

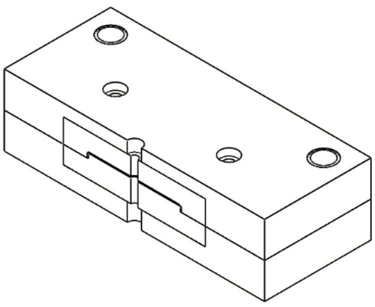
Tolerance designing of mold must consider the properties of the molding materials. Typical mold tolerances can refer to the data reported by the Society of the Plastics Industry, a U.S.-based trade association that's now known as the Plastics Industry Association (PIA, or PLASTICS). For example, the dimensional tolerances of mold for various materials are listed in **Table 1**. These values provide the general guidelines in mold designing for machining.

CASE STUDY

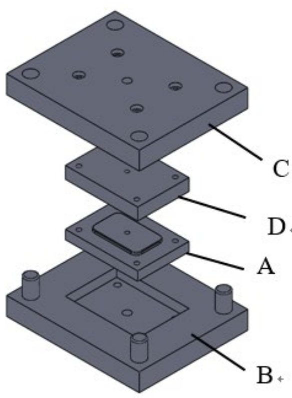
A plastic injection mold mainly is consisted of two molds, the protrusion half (the male mold) and the cavity half (the female mold). The components of the molds are typically machined from hardened steel, aluminum alloy, and/or beryllium-copper alloy, etc.

Mold Tolerances

The structure of a mold usually is consisted of the mold cores and the mold plates including the injection and ejection devices. The mold cores are designed according to the shapes and the dimensions of the injected products. The mold cores are assembled with the mold plates which are fixed on the mold bases to form the male and female molds. The male and female molds must be aligned accurately when closing for ensuring the smooth of the connection face of the injected products. The aligning of the male and female molds always is driven through the guide post and the guide sleeve which are fixed on the plates.



(a) The assembly diagram,

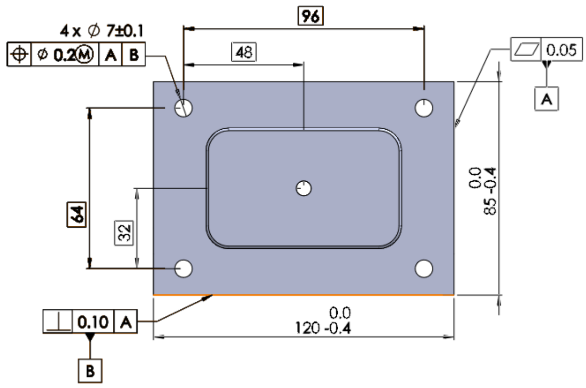


(b) The explosion diagram

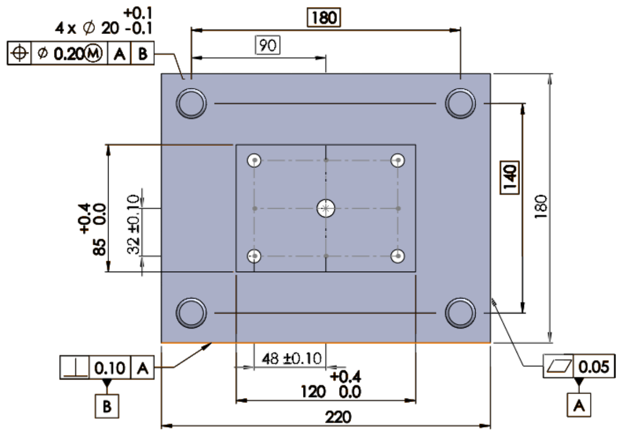
Figure 7. The primary parts (the original design)

A design (original) of the main parts of a mold is shown in **Figure 7**. The mold is consisted of four parts, the male mold core (A), the male mold plate (B), the female mold plate (C), and the female mold core (D).

The dimensions of the parts corresponding to fitness are designed with one-way tolerance to avoid assembly interference. The position sizes are expressed with geometric tolerances marked with the ideal dimensions plus two-way tolerances. The designs for the dimensions and the tolerances of the mold are shown in **Figure 8** and **Figure 9**. The injected product is a thin rectangular box in Figure 7, Figure 8, Figure 9 and Figure 10.

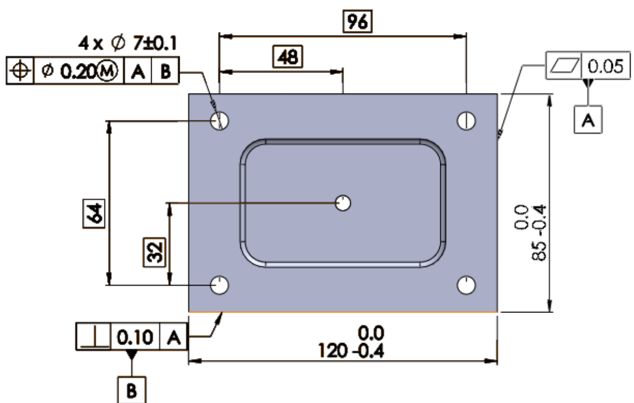


(a) The male mold core (A),

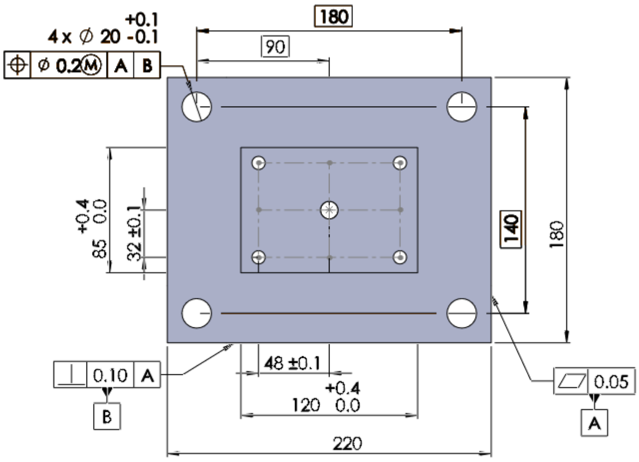


(b) The male mold plate (B)

Figure 8. The sizes of the male mold



(a) The female mold core (D),



(b) The female mold plate (C)
Figure 9. The sizes of the female mold

Mold Tolerance Analysis

The mold cores are fixed on the mole plates with bolts so that the assembly has a tolerance stack-up. The combinations of the guide posts (sleeves) and the mold plates always are designed with tight fits so that the tolerance stack-up can be ignored. The precision of a mold can be expressed with the assembly deviations of the male mold and the female mold. As a result, the dimensional chain is constructed according to the assembled relationship of the mold as shown in **Figure 10**. In Figure 10, it shows the cross section of an injected product.

The deviation (X) of the central lines of the male-female molds can be calculated by the equation

$$X = -A1 + B1 - B2 + C1 - C2 + D1 \qquad (5)$$

where A1, (B1, B2), (C1, C2) and D1 are the related dimensions of the male mold core, male mold plate, female mold plate, female mold core, respectively.

The tolerance analysis of the mold is done according to the dimensions of the mold and the dimensional chains. The dimensions and the tolerances of the features and the total tolerances in WC and RSS are shown in **Table 2**. The total tolerances of WC and RSS are calculated from Eq.(2) and Eq.(4), respectively.

The bonus tolerances (BT) are generated by the maximum material condition (MMC), BT=(maximum diameter-MMC)/2, such as A1 Bonus=(7.1-6.9)/2=0.1 mm, B2 Bonus=(20.1-19.9)/2=0.1 mm. The assembly tolerance (B-C) is set to 0 because the male mold plate (B) and the female mold plate (C) are driven by the guide posts and the sleeves which usually are precisely aligned in mold machining. The total tolerances of the mold in WC and RSS are ±1 mm and ±0.237 mm by tolerance calculations, respectively. The Cpk indicating the quality levels of the manufacturing tolerances corresponding to the specifications. The quality level is excellent while the Cpk are between 1.33 and 1.66. This paper takes the minimum Cpk value 1.33 to calculate the statistical tolerances.

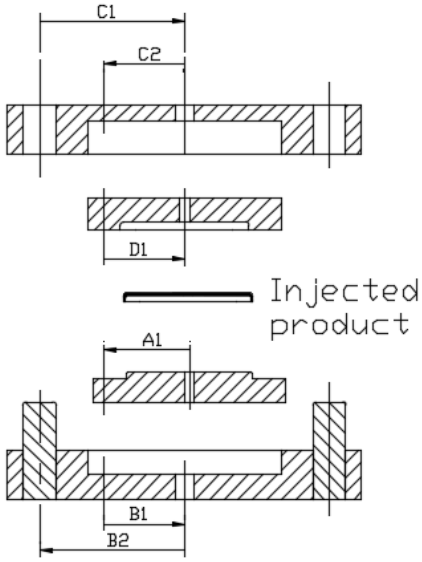


Figure 10. Dimensional chains of the mold

The total tolerances in RSS represent the 3σ tolerances in manufacturing. The probabilistic distribution of the total tolerances (RSS) can be expressed as normal distribution as shown in **Figure 11**. The red areas are the portions that the total tolerances exceed the intervals of the precision requirements, i.e. the unqualified rates.

Table 2. Tolerance analysis (the original design)

					C_{pk}=1.33	Worst Case			Staistic(RSS)		
Name	Direction (V)	Specification			Dimension	Symmetry	Min	Max	3 sigma (99.73%)		
		Dim	Tol(-)	Tol(+)	Mean	Tol(+/-)			Range (+/-)	Min	Max
A1	-1	48	0.1	0.1	48	0.1	47.9	48.1	0.075	47.925	48.075
A1 Bonus	1	0	0.1	0.1	0	0.1	-0.1	0.1	0.075	-0.075	0.075
B1	1	48	0.1	0.1	48	0.1	47.9	48.1	0.075	47.925	48.075
B2	-1	90	0.1	0.1	90	0.1	89.9	90.1	0.075	89.925	90.075
B2 Bonus	1	0	0.1	0.1	0	0.1	-0.1	0.1	0.075	-0.075	0.075
Assembly(B-C)		0	0	0	0	0	0	0	0.000	0.000	0.000
C1	1	90	0.1	0.1	90	0.1	89.9	90.1	0.075	89.925	90.075
C1 Bonus	1	0	0.1	0.1	0	0.1	-0.1	0.1	0.075	-0.075	0.075
C2	-1	48	0.1	0.1	48	0.1	47.9	48.1	0.075	47.925	48.075
D1	1	48	0.1	0.1	48	0.1	47.9	48.1	0.075	47.925	48.075
D1 Bonus	1	0	0.1	0.1	0	0.1	-0.1	0.1	0.075	-0.075	0.075
		0				**1**	**-1**	**1**	**0.237**	**-0.237**	**0.237**

The values can be obtained by computing the probabilities of the tolerance distribution. In this case study, the unqualified rates would be 5.76% if the tolerances are ±0.1 mm and the precision requirements of the mold are ±0.15 mm.

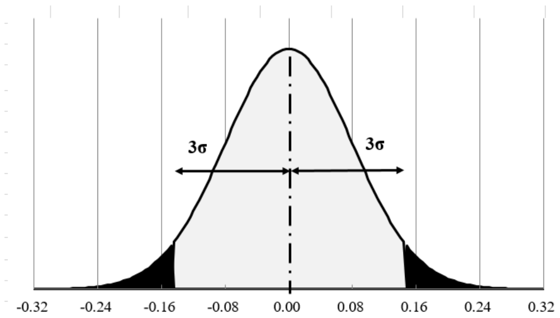


Figure 11. Probability distribution of the total tolerances

The unqualified rates of the mold under different tolerance designs and precision requirements can also be estimated as listed in **Table 3**. For instance, if the part tolerances are designed with ±0.05 mm, the total tolerance of the mold in RSS will be ±0.121 mm, and the unqualified rate will be 21.51% when the precision requirements being ±0.05 mm. If the precision requirements are enlarged to ±0.1 mm, the unqualified rate will be 1.32%.

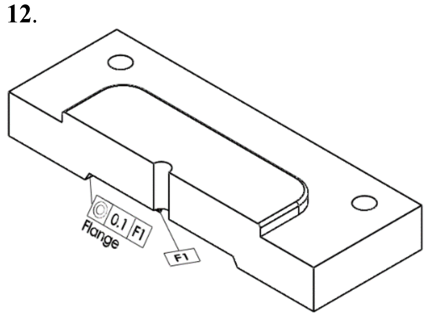
The evaluations of the unqualified rates can be obtained by using excel software with normal distribution function, Normdist(x, averg, std, True), to calculate normal distribution of 3σ. The unqualified rate is shown as the black areas in **Figure 11**, then, [1-Normdist(-x, averg, std, True) represents the black area of right side, and Normdist(-x, averg, std, True) represent the black area of left side. Then, the two black areas are added, the unqualified rate can be obtained. For example, in **Table 3**, if the design tolerances are ±0.1 mm and the precision requirements are ±0.15 mm, the ranges of statistic(RSS) are ±0.237 mm, the obtained unqualified rates of P(0.1, 0.15) is that P(0.1, 0,15)= 1-Normdist(0.15, 0, 0.237/3, 1) + Normdist(-0.15, 0, 0.237/3, 1) = 0.0576 =5.76%. In **Table 3**, using the same calculation processes, if the design tolerances are ±0.08 mm, ±0.05 mm, ±0.02 mm, and the ranges of 3σ for statistic(RSS) are ±0.194 mm, ±0.121 mm, ±0.049 mm, respectively, the precision requirements are ±0.05 mm, ±0.1 mm, ±0.015 mm, the unqualified rates can be obtained, respectively.

Table 3. The unqualified rates (the original design)

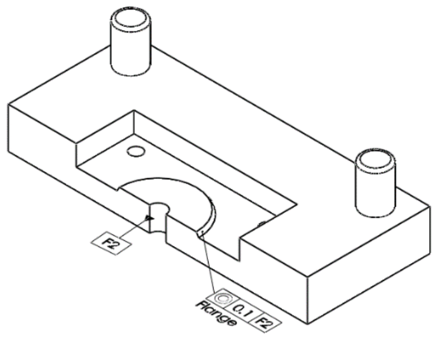
Design	RSS	Requirement		
Tol (+/-)	3σ	0.05	0.1	0.15
0.1	0.237	52.68%	20.56%	5.76%
0.08	0.194	43.94%	12.20%	2.04%
0.05	0.121	21.51%	1.32%	0.02%
0.02	0.049	0.22%	0	0

Design Improvements

A conical convex (concave) flange is designed at the center of the mold plate (core) for assembly positioning. The conical flange designs enable the mold core and the mold plate having the function of automatic centering when they assembled. The designs of the positioning flanges of the male mold and its tolerances marked with concentricity are shown in **Figure 12**.



(a) The mold core,



(b) The mold plate

Figure 12. The male mold of the new design

Table 4 Tolerance analysis of the new design

Name	Direction	Specification			C_{pk}=1.33 Dimension	Worst Case			Staistic(RSS) 3 sigma (99.73%)		
		Dim	Tol(-)	Tol(+)	Mean	Symmetry Tol(+/-)	Min	Max	Range (+/-)	Min	Max
A1	-1	30	0.1	0.1	30	0.1	29.9	30.1	0.075	29.925	30.075
Assembly (A-B)		0	0	0	0	0	0	0	0.000	0.000	0.000
B1	1	30	0.1	0.1	30	0.1	29.9	30.1	0.075	29.925	30.075
B2	-1	90	0.1	0.1	90	0.1	89.9	90.1	0.075	89.925	90.075
B2 Bonus	1	0	0.1	0.1	0	0.1	-0.1	0.1	0.075	-0.075	0.075
Assembly (B-C)		0	0	0	0	0	0	0	0.000	0.000	0.000
C1	1	90	0.1	0.1	90	0.1	89.9	90.1	0.075	89.925	90.075
C1 Bonus	1	0	0.1	0.1	0	0.1	-0.1	0.1	0.075	-0.075	0.075
C2	-1	30	0.1	0.1	30	0.1	29.9	30.1	0.075	29.925	30.075
Assembly (C-D)		0	0	0	0	0	0	0	0.000	0.000	0.000
D1	1	30	0.1	0.1	30	0.1	29.9	30.1	0.075	29.925	30.075
		0				**0.8**	**-0.8**	**0.8**	**0.212**	**-0.212**	**0.212**

The new design changes the positioning type of the original design so that the dimensional chain is modified according to the connection relationship as shown in **Figure 13**.

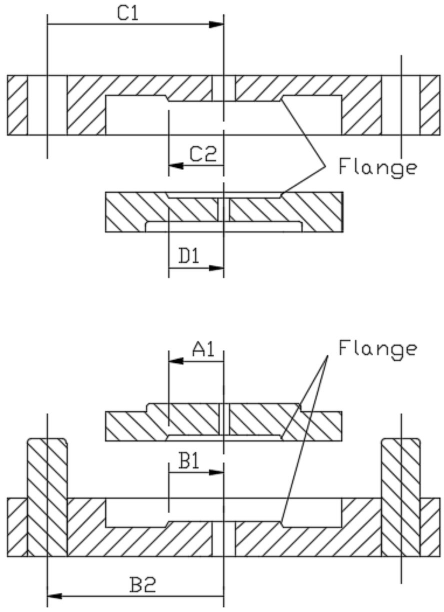


Figure 13. Dimensional chains of the new design

The tolerances of the conical flanges (A1, B1), (C2, D1) are designed with concentricity of the central holes (0.1 mm). The tolerance analysis of the new design is shown in **Table 4**. The value of C_{pk} is taken as 1.33 in **Table 4** and this C_{pk} value is the same in **Table 2** in order to make comparison each others. There are two paired assemblies (A, B) and (C, D) are added and two tolerances (A1 Bonus, D1 Bonus) are removed because of the positioning function of the bolt holes are replaced by the conical flanges. By using the similar calculation processes as shown in Table 2, the results of the analysis for the new design can be obtained. The total tolerances of the new design in WC and RSS are ±0.8 mm and ±0.212 mm, respectively. The total tolerances are smaller than those of the original design. It implies that the mold precision is promoted.

The results of the unqualified rates of the new design under the same tolerance designs and precision requirements in **Table 3** are listed in **Table 5** by using the similar calculation process.

Table 5. The unqualified rates of the new design.

Design	RSS	Requirement		
Tol (+/-)	3σ	0.05	0.1	0.15
0.1	0.212	47.92%	15.70%	**3.38%**
0.08	0.174	38.86%	8.47%	0.97%
0.05	0.109	16.88%	0.59%	36ppm
0.02	0.044	0.07%	0	0

The results show that the stacked tolerances can be decreased through the design modification.

The quality improvements of a design can be measured by evaluate the increase of the acceptance rates as well as the decrease of the unqualified rates. It is defined as

$$R = \frac{P_f^o - P_f^r}{1 - P_f^o} \tag{6}$$

where P_f^o and P_f^r are the unqualified rates of the mold in the original and new designs, respectively. The quality improvements of the mold in various tolerance designs are listed in **Table 6**. The unqualified rate of the new design is improved to 3.38 % from 5.76 % of the original design for part tolerances ±0.1 mm. The quality improvements (R) of the mold is 2.53 % as shown in **Table 6**. The improved spaces get smaller while the tolerance ranges of the parts reduced. This is because the parts designed with the more narrow tolerances, the smaller the unqualified rates of the mold (the higher the quality levels) would be. It denotes that the tolerance improvements of a poor design are bigger than a good design by the proposed method.

Table 6 The quality improvements

Items	Unqualified rates			
Part Tol (+/-)	0.1	0.08	0.05	0.02
Qriginal	5.76%	2.04%	0.02%	0
New	3.38%	0.97%	0.0036%	0
Improvement (R)	2.53%	1.09%	0.02%	*
Mold tolerance requirements (+/-) 0.15 mm				

In engineering design, setting proper tolerances for the components is a critical job in satisfying the functional requirements and the manufacturing costs. Through the tolerance analysis, the total tolerance of an assembly can be calculated, further, the unqualified rates of a product under different tolerance designs can be reasonably estimated. It offers a systematic method to predict the qualities of a design by tolerance analysis as well as developing tolerance-based reliability design for a product.

CONCLUSION

This paper reports the methods of reducing tolerance stack-up for an injection mold. The general methods of tolerance marking are introduced including the influence of correction symbols of geometric tolerance to the tolerances. The often-used methods in tolerance analysis and tolerance allocation are introduced to perform tolerance analysis and tolerance designing. A positioning method for parts assembling is proposed to reduce the stacked tolerances. The assembly tolerances and the unqualified rates of a mold

under different part tolerances are analyzed to identify the feasibility of the proposed method. The analyzed results show that the total tolerances are reduced from 0.237 mm to 0.212 mm and the unqualified rates are from 5.76% to 3.38% under the same tolerance designs. The quality improvement of the mold is 2.53% by the proposed method in reducing tolerance stack-up. This study is helpful in developing tolerance-based reliability design and quality improvement for engineering designing.

REFERENCE

ASME Y14.5, Dimensioning and tolerancing (2009).

Bacharoudis, K., A. Popov and S. Ratchev, "Tolerance allocation: A reliability based optimisation approach", Procedia Manufacturing, Vol.51, pp.1038-1045 (2020).

Bjorke, O., "Computer Aided Tolerancing, (2nd Edition)", New York : ASME Press, University of Trondheim, Norway, (1989)

Chase, K. W., "Tolerance allocation methods for designers", Brigham Young University (1999).

Cao, Y., T. Liu, and J. Yang, "A comprehensive review of tolerance analysis models", International Journal of Advanced Manufacturing Technology, Vol.97, pp.3055–3085 (2018).

Fischer, B. R., "Mechanical Tolerance Stackup and Analysis, (2nd Edition)", CRC Press, Taylor &Francis Group (2011).

Huang, X. Y., S. H. Yang, B. N. Zou and M. C. Tsai, "Tolerance estimation and machining error monitoring for smart manufacturing", Machinery Industry, Vol.425, pp.60-66 (2018).

ISO 286-1, "Geometrical Product Specifications - ISO code system for tolerances on linear sizes - Part 1: Basis of tolerances, deviations and fits" (2010).

Otsuka, A. and F. Nagata, "Stack-up analysis of statistical tolerance indices for linear function model using Monte Carlo simulation", International Conference On Engineering Design, ICED15, Politecnico Di Milano, Italy (2015).

Sahani, A. K., P. K. Jain, S. C. Sharma and J. K. Bajpai, "Design verification through tolerance stack-up analysis of mechanical assembly and least cost tolerance allocation", Procedia Materials Science, Vol.6, pp.284-295 (2014).

Tsai, J. C. and S. R. Wu, "Tolerance Analysis and Re-distribution of A Machine Tool Spindle by Negative Tolerancing", Procedia CIRP, Vol.10, pp.267-270 (2013).

Tsai, J. C., F. C. Chen and J. H. Dai, "Reduction of Tolerance Stack-up by Grouped Random Assembly for Components with Uniform Distributions", Procedia CIRP, Vol.27, pp.260-263 (2015).

Tsai, Y.T., K. H. Lin and Y.Y. Hsu, "Reliability design optimisation for practical applications based on modelling processes", Journal of Engineering Design, Vol.24(12), pp.849-863 (2013).

Tsai, Y. T. and C. S. Chen, "Research on virtual-real integration in injection molding parameter optimization", Insight Machinery Magazine, Vol.7(12), pp.30-41 (2022).

Tsai, Y. T. and K. H. Lin and C. S. Chen, "A study of tolerance allocation and stack-up analysis to improve the assembly precision of an injection mold", Journal of the Chinese Institute of Engineers, Vol.46(5), pp.479–489 (2023).

一個減少模具零件組裝公差堆積的方法

蔡有藤
宏國德霖科技大學機械系

陳春生
曜碁企業股份有限公司

摘要

公差設計將決定產品的精度及後續的製造成本，設計時配置適當的公差以滿足產品功能要求至為重要，本文介紹幾何公差標註及公差分析方法，研究定位公差修正符號對公差影響，建立模具組裝尺寸鏈以進行公差分析和設計改良。本研究提出了一種組裝定位方法，以減少公差推積，提高組裝精度，利用製造自然公差和統計公差來估計組裝不合格率，分析模具在不同公差設計下的不合格率；研究結果顯示，透過定位設計可降低組裝公差，在相同精度要求和公差設計，利用本文所提方法可使模具總公差的不合格率由 5.76%降到 3.38%，本研究提供公差設計和分析的系統性方法，確保公差設計符合功能要求。