DESIGN

for

FIT

Applications of Geometric Tolerancing
to Machine Design

First Edition  Faryar Etesami
Design for Fit
Applications of Geometric Tolerancing to Machine Design
First Edition

Faryar Etesami
Mechanical and Materials Engineering Department
Portland State University

D4F Publishing
P.O. Box 1651, Tualatin, Oregon 97062
Preface

This book is mainly written for students enrolled in an undergraduate program in mechanical engineering (BSME) or similar programs. The material presented is based on my notes for teaching mechanical tolerancing for nearly thirty years. The book’s emphasis is on fit requirements for machine components. Fit assurance makes up the majority of challenging applications in tolerancing. Design for specific functions is easy by comparison. For design engineers, knowing how to apply geometric tolerances correctly has been a challenge even for engineers who have practiced geometric tolerancing for a long time. The syntax and meaning of geometric tolerancing statements can be learned easily and quickly but knowing how to use them correctly as a design engineer is much more difficult. For years I taught the geometric tolerancing standards in great detail as one may teach a foreign language using a dictionary. That method of training was fine for those on manufacturing or inspection career paths but was not very useful to design engineers. Finally, I decided to teach subject not as a geometric tolerancing class but as a design-for-fit class. I found this new method to be far more effective than just covering every detail of the GDT standard.

The objective of this book is to present the subject of design-for-fit in a format that is suitable as a college-level course. I believe people learn in the context of simple examples free of distractions. They are then able to apply what they learned in the most complex situations. For that reason, I have done my best to convey the concepts of design for fit in the context of simple examples. I believe that a graduating mechanical engineer who designs and documents parts should have a basic understanding of the design-for-fit principles presented in Chapters 1-9. The book can be used to support a junior-level or senior-level class in a BSME program or it can make a companion textbook for the capstone design classes. Interested students or practitioners can then follow up with the more advanced topics in Chapter-10 and Chapter-11 on multipart fits and fit analysis. Since this book does not cover every detail of the geometric tolerancing standard I recommend having a copy of the standard document as an official and legal reference. Low-cost pdf versions of the GDT standard can be found online.

Faryar Etesami
## CONTENTS

### Chapter-1

**Dimensions and Tolerances**

1.1. Overview .................................................................................................................. 5
1.2. Evolution of tolerancing .......................................................................................... 6
1.3. Geometric versus dimensional tolerances .............................................................. 10
1.4. Importance of clear geometry communication practices ................................. 15
1.5. Functional feature vocabulary .............................................................................. 16
1.6. Preferred practices in dimensioning ................................................................. 20
1.7. Figures of accuracy for dimensions and tolerances ........................................ 38

### Chapter-2

**A Design Engineer’s Overview of Tolerance Statements - Part-I**

2.1. Overview .................................................................................................................. 42
2.2. Interpretation of geometric tolerances .................................................................. 42
2.3. Dimensional tolerances ......................................................................................... 44
2.4. Geometric tolerances .............................................................................................. 45
2.5. Tolerances of form .................................................................................................. 47
2.6. Flatness ...................................................................................................................... 47
2.7. Meaning of $\phi$ symbol next to the tolerance value ........................................... 50
2.8. Straightness ............................................................................................................ 52
2.9. Circularity ................................................................................................................ 53
2.10. Cylindricity ............................................................................................................ 55
2.11. Tolerances of orientation ..................................................................................... 56
2.12. Parallelism .............................................................................................................. 58
2.13. Identifying the axis of an imperfect cylinder ...................................................... 60
2.14. Perpendicularity .................................................................................................... 64
2.15. Angularity ............................................................................................................... 67
Chapter-3
A Design Engineer’s Overview of
Tolerance Statements - Part-II................................. 74

3.1. Overview ........................................................................................................... 74
3.2. Tolerances of location ......................................................................................... 74
3.3. Position tolerance ............................................................................................... 74
3.4. Meaning of modifier applied to datum features .................................................. 88
3.5. Composite position tolerance – multiple segments ............................................ 90
3.6. Composite position tolerance – single segment .................................................. 94
3.7. Runout tolerances ............................................................................................. 96
3.8. Profile tolerances ............................................................................................. 105
3.9. Profile tolerances in place of ± tolerances ....................................................... 109

Chapter-4
Default Tolerances ................................................................. 125

4.1. Overview ........................................................................................................... 125
4.2. Default tolerancing methods .............................................................................. 125
4.3. Datum targets ................................................................................................... 130
4.4. Default tolerancing and small features ............................................................ 134
4.5. Default tolerances for fillets and rounds .......................................................... 135
4.6. A different way of dealing with small features ............................................... 136

Chapter-5
Tolerance Design for Unconstrained Fits Between Two Parts
Part-I ................................................................................................. 147

5.1. Overview ........................................................................................................... 147
5.2. Unconstrained fits versus constrained fits ......................................................... 147
5.3. Fit assurance between two parts .......................................................................... 151
5.4. Theoretical gages for tolerance statements that use the symbol ......................... 157
5.5. Fit formula ....................................................................................................... 158
5.6. Default form control implied by limits of size .................................................. 161
5.7. Use of zero geometric tolerance at MMC ......................................................... 162
5.8. Fit formula when using zero geometric tolerancing at MMC .............................. 164
5.9. Defining a fit boundary without the modifier ...................................................... 164
5.10. A note on the use of modifiers ....................................................................... 165
Chapter-6
Tolerance Design for Unconstrained Fits Between Two Parts
Part-II .................................................................173
6.1. Overview ........................................................................173
6.2. An overview of the limits and fits standard (ANSI B4.2) .................................................. 175
6.3. Preferred sizes ................................................................177
6.4. Offsets letters (fundamental deviations indicator letters) .................................................. 178
6.5. International tolerance grades .................................................. 180
6.6. Preferred fits ..................................................................180
6.7. Dependency of offsets and tolerance grades on basic size .............................................. 191
6.8. Tolerance design examples in precision fit applications ........................................... 191

Chapter-7
Tolerance Design for Orientation-constrained Fits
Between Two Parts ..................................................................205
7.1. Overview ........................................................................205
7.2. The fit boundary and the theoretical gage .................................................................. 207
7.3. A word on inspection ................................................................213
7.4. MMC boundary as fit boundary ........................................................................ 214
7.5. Situations where MMC modified tolerances cannot be used .................................... 216
7.6. Datum feature flatness ......................................................................217
7.7. Is it necessary to specify axis straightness? ................................................................ 222
7.8. Defining a fit boundary without the $\oplus$ modifier .................................................. 224

Chapter-8
Tolerance Design for Location-constrained Fits
Between Two Parts ..................................................................227
8.1. Overview.........................................................................227
8.2. The fit boundary and the theoretical gage .................................................................. 228
8.3. Datum priority ...................................................................233
8.4. Datum frame precision ..................................................................238
8.5. Imperfect geometry relationships are not intuitive .................................................. 240
Chapter 1
Dimensions and Tolerances

1.1. Overview

This chapter presents an introduction to tolerancing practice and suggests guidelines for clear graphics communication. After a part’s exact geometry is defined a design engineer must also specify the part’s geometric tolerances. Tolerances define allowable deviations of part features from their theoretical definitions. Since manufacturing variations are inevitable, tolerancing information allows production personnel to select suitable processes and methods to meet part precision requirements at minimum cost.

Tolerancing specifications include tolerance types and tolerance values. For example, if a surface is to be flat with high precision, then the designer applies a standard flatness tolerance symbol and specifies a tolerance value. The smaller the tolerance value, the more flat the surface would be manufactured, but usually at a higher cost. The geometric tolerancing standard to which the material in this book refers to is the ASME Y14.5-2009 stands, which from now on, will be referred to as the GDT standard. The GDT standard provides a variety of symbols a designer can use to control various geometric aspects of a part.

The design engineer’s challenge is to create a set of precision specifications that is just enough to assure proper fit and function of a part in an assembly. The ability to create well-toleranced parts is an important skill for a mechanical design engineer. Improper use of tolerance types or tolerance values can affect the cost of production as well as the fit and function of assemblies. To be safe, many designers resort to over-specification of tolerances. Tolerance types that are not needed or tolerance values that call for unnecessary precision would lead to higher production costs and delays in a variety of ways such as:

- The need for more expensive processes or machines
- The need for additional or more time-consuming processing steps
- Lack of in-house capability
- The need for more expensive tooling
- A reduction in throughput
- The need for more expensive inspection tools
- An increase in rework and reject rates
- Excessive lead-times

On the other hand, lack of necessary tolerance types or tolerance values that do not call for enough precision reduce the cost of production at the risk of non-assembly or functional compromises. Some of the more common failures associated with lack of sufficient geometric precision are:

- Risk of parts not fitting properly into an assembly
- Improper performance
• Improper sealing
• Loss of operation accuracy
• Loss of power
• Loss of functionally critical alignments
• Improper thermal or electrical conduction

• Structural Failures
  • Improper load distribution
  • Unexpected dynamic loads
  • Unexpected loading modes
  • Unacceptable changes of stiffness

• Reduced service life
  • Excessive friction or wear
  • Excessive noise
  • Excessive heat
  • Excessive vibration

The great majority of tolerancing needs for typical parts relate to their fit requirements or their alignment requirements necessary for fit. For that reason, this book primarily discusses tolerancing needs for proper fit.

Like other engineering specifications, good tolerancing is not merely about what works but what works at minimum cost. Geometric tolerances have a big impact on the final production costs.

1.2. Evolution of tolerancing

Before the industrial revolution, craftsmen often made the necessary parts and fit them together in their shops. Each part was custom made to fit where it needed to fit. It was neither necessary nor wise to make all the parts first and hope they would fit together properly at the end. The practice of build-and-fit was possible because the entire assembly was made by the same skilled workers in the same shop. Customers simply expected the entire assembly to work as described with some desired overall dimensions. Even today prototypes are often made this way because this method requires less precision work than building all the parts first and fitting them together later. It is also easier and quicker for the design engineers to leave all the details to the manufacturing personnel.

Mass production made the build-and-fit scheme impractical. Parts needed to be made on different machines and task specialization allowed high throughputs. Other skilled workers or machines put the parts together forming the final assemblies. The necessity of faster product introduction also resulted in specialization of skills. Shape design became the responsibility of the designers, draftsmen assisted designers by creating detailed drawings, production specialists designed the tooling and produced parts most efficiently, and quality control specialists checked the accuracy of the parts. Drafting guidelines were introduced to standardize the graphical communication of geometric information between designers, manufacturing, and quality control people.
To make sure that the parts were made with sufficient accuracy to fit together during assembly, design engineers had functional gages built and sent with their drawings that described the part dimensions. Each functional feature of a part was to be checked with two types of physical gages, a GO gage and a NOT GO gage. A part that would fit the GO gage was assured to fit a mating part in the assembly. The GO gage, sometimes made of wood, simply represented the worst-case geometry of a mating part for fit purposes. Figure 1-1 shows a portion of a chainsaw assembly.

![Figure 1-1. A portion of a chainsaw assembly](image)

Figure 1-2 shows a chainsaw link with its two critical holes along with a GO gage which is a flat plate with two pins. GO gages simulate the contacting surfaces and fitting features of a mating part.
Figure 1-2. The chainsaw link and a GO gage checking its fit to a mating part

GO gages check the fit of a feature into assembly but they do not check for excessive play in the fit when the holes are too large. NOT GO gages were used to make sure the holes were not too large. A hole that would go through a NOT GO gage was too large leading to a loose fit and creating too much slack or play to be functionally acceptable. The NOT GO gage for the chain link is a plug gage that is used individually on each hole. The chain link and the plug gage are shown in Figure 1-3.
An acceptable part feature would clear the GO gage, assuring fit, but not the NOT GO gage, assuring acceptable play and proper function. Other non-critical features of parts were assumed to be acceptable as made when the tools and workmanship conformed to established standards of good manufacturing. No gages were made for non-critical features.

Building and sending physical gages with part drawings were difficult to do in a design environment. In later years, design engineers simply created separate drawings necessary to build the gages at the manufacturing site with some added precision instructions. The precision instructions, or tolerancing instructions, were in the form of ± variation limits on the dimensions of the gages to be made. With this additional description of accuracy, the gages could be built at the manufacturing site where the parts were being made. Eventually, these ± limits were directly applied to the part dimensions from which appropriate gages could be made to verify the fit and function of precision features. This eliminated the need for the designers to create multiple sets of drawings, one for the part and others for the gages. It also became easy to simply check the dimensions of the part for conformance. Unfortunately, this practice led to serious unintended consequences.
Applying dimension limits by designers and checking dimensions by inspectors became the norm in tolerancing until it was replaced by geometric tolerancing practices that once again introduced the concepts of functional gaging back into the tolerancing standards and tolerancing practice.
1.3. **Geometric versus dimensional tolerances**

At the time of its introduction, the geometric variation control based on ± limits on part dimensions and angles were logical extensions of dimensioning, easy to apply, easy to understand, and easy to verify. This is true only when the geometry of the part remains free of deformations and distortions. Deformations and distortions make it difficult to interpret the meaning of dimension limits. As more knowledge was gained regarding the geometric requirements of fit and function, it became clear that dimensional tolerances alone did not provide the best specification toolbox for assuring fit and function.

The concept of gaging and gage description is at the heart of what constitutes geometric tolerancing. A simple example can highlight the advantage of geometric tolerancing over just using dimension limits. Consider rolling a heavy, tall refrigerator on casters past a doorway. The width of the refrigerator is just below 835 mm and the width of the doorway is just over 835 mm. There is a good chance, however, that the refrigerator would not fit through. Figure 1-4 highlights the point.

![Figure 1-4. The width of the refrigerator alone may not be sufficient to assure fit](image)

On the left figure, the refrigerator and the doorway are shown to be free from all distortions. In reality, however, this never happens. On the right figure the refrigerator is tilted slightly to the left with respect to the plane of casters, while the doorway is assumed to be free of distortions. The resulting arrangement leads to interference. If the doorway is also tilted, it can make the situation even worse.

You can imagine how specifying the width dimension limits alone can put fit ability in jeopardy. The designer who wants to assure fit using dimension limits must also specify or explain to measure the width correctly as shown in Figure 1-5.
Of the three methods of measurement shown, only the 840 mm value is functionally correct for fit. The other two measurements, albeit both reasonable measurements of the “width” of the refrigerator, lead to incorrect values for fit. A tolerancing scheme based on dimension limits simply ignores the reality that shapes can have deformations and distortions. You may describe the required method of measurement by notes but the GDT standard provides an elegant way of describing exactly such a requirement using a compact symbolic notation.

By making gages, the dimensional measurement confusion would never happen. The gage for the refrigerator would be a perfect 835 mm doorway as high as the refrigerator and made nearly perfectly with gage-making tolerances. If the refrigerator passes through this gage properly, it would also pass through the real doorway. Note that the gage geometry description by itself is not sufficient to assure a correct fit. The gaging instructions are necessary to ensure the refrigerator passes through the gage while casters are in contact with the floor. The GDT standard provides a set of symbols to easily describe the gages and the gaging methods for this and other common fit applications. The strength of the GDT standard specifications is that it allows a designer to specify the precision needs of a part by following and mimicking the way a part is intended to fit into an assembly.
Unlike dimensional tolerances, geometric tolerancing statements do not ask for any numerical measurements at all. They simply describe the geometry of the necessary gages to assure fit and function. Of course an inspector who does not want to build a gage, for cost or practical reasons, is forced to make the necessary measurements to be able to judge with confidence whether the feature would pass such a gage if it had been made. In the case of the refrigerator, the inspector may easily measure the width of 830 mm and also measure the angle of tilt. Combing the two measurements would come close to the functional dimension of the refrigerator.

In design, the basic elements of function are features. Features are collections of surfaces that, as a whole, can be associated with a particular function. This is similar to the use of language in which the basic elements of communication are words not individual letters. A feature-based geometric specification language directly deals with features and their functions. The geometric tolerancing standard provides this feature-based language and a gage-based methodology for geometry control. Geometric tolerancing allows the design engineers to separately control various geometry aspects of features when needed. Such aspects include feature size, form, orientation, and location.

Today, the preferred method of specifying acceptable manufacturing tolerances is a combination of geometric and dimensional tolerances. Dimensional tolerances are mainly used to describe size limits of cylindrical features or width features (slots and rails). Figure 1-6, for example, shows both the dimensional and the geometric tolerancing methods of controlling the size and the distance between two holes. The geometric method will be explained later in detail.
Figure 1-6. Specifying the tolerance on the distance between two holes using dimensional tolerancing (top) and geometric tolerancing methods (bottom)

The two schemes both control the size and distance between the two holes but they do not have the same meaning. The specification shown in the bottom figure is functionally more meaningful and preferred to the method shown on the top figure. The details will be presented in later chapters.

Figure 1-7 shows the dimensions necessary to define the geometry of the chainsaw link previously discussed. In a tolerancing scheme based on ± limits on dimensions, each dimension needs to have an associated tolerance either explicitly or through default tolerances in order to create a production drawing. The chainsaw link requires a lot of ± tolerances to be fully tolerated. Except for the plate thickness tolerance, these tolerances are not shown on this figure because the drawing becomes difficult to read.
Figure 1-7. The chainsaw link dimensions

Figure 1-8, on the other hand, is a complete tolerancing scheme using feature-based geometric tolerancing specifications. Notice that only two dimensions are tolerated using ± values. The rest of the dimensions are not shown because they are theoretical design dimensions and available in the CAD model of the part and can be automatically inserted into the drawing. The details of this tolerancing process will be presented in the later chapters but the fact that a design engineer need not break down features into surfaces and pay attention to every single dimension of a part is a significant time saver.
Geometric tolerancing promotes a feature-based and functional approach to tolerancing and allows a design engineer to control feature size, form, orientation, and location with varying degrees of precision as needed for fit and function. For example, in a feature-based way of thinking about tolerances, the chainsaw link has only three functional features. The first feature is the entire outer profile and is controlled with a single tolerance statement with low precision. The second feature is the pattern of two holes controlled by another geometric tolerance statement at high precision. The third feature is the plate thickness controlled by a dimensional tolerance at medium precision.

1.4. Importance of clear geometry communication practices

Design engineers communicate the theoretical shape of parts to manufacturing through dimensioning. While the complete geometry of a part can be accessed through its 3D CAD model, the use of 2D drawings is still quite common for geometry communication. Poorly dimensioned parts can lead to incorrect geometry interpretations or delays due to the need to clarify vague or missing information. Part drawings are not just to communicate information to manufacturing as design engineers often discuss the merits of different design alternatives using 2D dimensioned drawings as well. Poorly dimensioned parts reduce the effectiveness of design work as well. Today, the old engineering graphics courses are quickly disappearing as a part of the mechanical engineering curricula, replaced by 3D solid modeling classes. The lack of basic graphics
training has led to new design engineer’s inability to create well-dimensioned parts. Improper dimensioning often leads to frequent conflicts with manufacturing personnel.

A well-dimensioned part drawing is like a well-written report. Good writing is about conforming to good writing principles that give a reader easy access to the information presented in a report. Likewise, a well-dimensioned part drawing presents the geometry of all the part features in a format that is easy to understand completely and quickly. As a badly written report is often an indication of poorly-understood ideas, a badly dimensioned part drawing is often a sign of poorly-understood fit and function needs.

To avoid unnecessary communication problems with manufacturing and others, design engineers should learn the simple rules of good dimensioning practice and develop the habit of delivering high-quality drawings to manufacturing and other design engineers. At first, conforming to good dimensioning rules may appear burdensome but with practice the methods become natural, intuitive, and easy to follow. The ability to conform to good dimensioning practices is also helpful in adhering to good tolerancing practices.

A lazy approach to dimensioning is to take a part drawing and randomly add dimensions to it until it is fully defined. This approach often leads to under-defined or over-defined parts with information scattered without any logic or organization. This method conveys little understanding of fit and function needs. A better approach to dimensioning is to recognize that each part is made up of a collection of features. Each feature (or feature pattern) can be described by a set of dimensions that define the shape of the feature itself and a set of dimensions that locate the feature with respect to other part features. In this respect, the word “feature” refers to a collection of part surfaces with a particular function. The function need not be precision as long as the feature has a particular purpose.

1.5. Functional feature vocabulary

It is helpful to use the proper feature vocabulary when dimensioning or communicating geometry information to other engineers. Figure 1-9 shows the typical features of a shaft in power transmission applications. Using a functional feature vocabulary helps thinking in terms of features and not just individual surfaces.
Figure 1-9. A transmission shaft with its associated functional features

Grooves and holes are often used with other identifiers such as snap-ring grooves or spring pin holes. Common features associated with fasteners are shown in Figure 1-10.

Figure 1-10. Various kinds of fastener-related features
Figure 1-11 shows some other common features on plate-like parts. A boss is a short raised surface of any shape. A profile is any combination of surfaces grouped together for functional and tolerancing purposes. Profiles can be internal (pockets) or external.

![Diagram of a plate-like part with labeled features: Profile, Boss, Rail, Slot, Slotted Hole.]

**Figure 1-11. Other common features of plate-like parts**

Figure 1-12 shows some of the common features of parts that usually fit onto rotating shafts.
Figure I-12. Common features of parts used with rotating shafts

The hub is the bulky part in the center of a wheel or a gear. The flange is the relatively thin disk attached to a hub. A D-hole is a hole with a flat face that fits into a D-shaped shaft for torque transmission. A lip is a small recess that fits a mating hole to create a fit with minimal play.

Figure 1-13 shows a shaft with several geometric features in addition to the main cylindrical shapes. A good approach to dimensioning this part is to recognize its functional features and completely dimension each feature’s internal geometry and location until no features remain. Thinking like manufacturing, a feature that is fully dimensioned is like a feature that is fully made. It is a good practice to check every feature of a part to make sure they are properly defined before sending the drawing to manufacturing. It is also a good practice to change your role and see the part as a virtual machinist who is going to make the part you designed. When dimensioning the part features, complete a feature’s dimensioning before going to the next one.
A neck is a groove between two different shaft diameters. The drawing and dimensioning for this part will be presented later in this chapter. The next section presents the necessary guidelines to create well-dimensioned parts.

### 1.6. Preferred practices in dimensioning

Most of the rules of dimensioning are developed to communicate complete geometry information clearly with minimal clutter. It is emphasized again that a drawing should not be viewed as a collection of lines and curves but as a collection of features. Once every feature is properly dimensioned, the dimensioning task is complete. The drawing reader should not only find all the necessary information to construct each feature of the part but find them easily and quickly where he/she expects them to be placed. If you have had no drafting training I suggest that you get a reference book on drafting or at least visit some web sites where more drafting rules and examples are presented.

In order to reduce the number of needed dimensions, the following geometric relationships are true in a part drawing. When no information is provided to indicate otherwise, the following relationships are implied:

- Lines that appear to be perpendicular are perpendicular.
- Lines that appear to be parallel are parallel.
- Lines that appear to be collinear are collinear.
- Lines and curves that appear to be tangent are tangent.
- Cylindrical features that appear to be coaxial are coaxial.

---

**Figure 1-13. A transmission shaft with its associated functional features**
Lines mentioned in the above rules also include the centerlines connecting the features of a pattern, such as a pattern of holes, whether such connecting lines are shown or not. The following geometric relationships are not implied when left unspecified.

- Lines that appear to have equal lengths are not implied to have equal lengths.
- Diameters and radii that appear to be equal are not implied to be equal.
- Features that appear to be symmetrically located with respect to a center plane or axis are not implied to be symmetrically located unless other information is provided.

Figure 1-14 illustrates some of the above rules. The dimensioning of this and other example parts shown in this section may not be complete to better illustrate certain key points.

![Figure 1-14](image.png)

Figure 1-14. Avoid double dimensioning. Lines that appear collinear are collinear.

The crossed out dimensions represent repeated dimensioning and should be removed. The 30 mm dimensions are both necessary as the appearance of equal length lines does not imply equal length lines. Similarly, the hole feature appearing in the middle of the part does not imply that it is in the middle and must be located both horizontally and vertically. If the 80 mm dimension is not shown, as in Figure 1-15, then the center plane of another feature should be shown to imply that the hole axis is aligned with the center plane of another feature. A geometric tolerance statement would also be necessary to
identify which feature the hole is aligned with as multiple features may qualify. The procedure for applying such tolerances will be presented in later chapters. It is worth noting that centering or alignment implies a dimension of zero which is not shown on a drawing. Therefore, in strictly dimensional tolerancing, there is no ability to indicate how precisely the features are to be centered. Fortunately, that is not a problem in geometric tolerancing. The side view also shows that the 40-mm slot is to be centered to the 60-mm outer rail feature. A tolerance statement is necessary to show the level of centering precision. If in the side view a dimension of 10 mm is shown then the center plane lines should be removed.

![Figure 1-15. Extended centerlines should be drawn to indicate that a feature is centered with respect to another feature](image)

A smaller scale 3D view is often added to a drawing but the drawing interpretation must not rely on such 3D images. Three dimensional models can be dimensioned but the practice is not uniform or widespread. In this book dimensions are not shown on 3D models or assembly drawings. This book uses the 3rd angle projection format which is commonly used in the United States.

Views that are not needed should be removed. You may insert additional views that you feel would be needed to correctly interpret the drawing or to provide additional dimensions. However, adding unnecessary views may reduce the readability of a drawing.

When creating drawings you are expected to adhere to other rules and preferences listed below. Not following these rules usually reduces the efficiency of communication. However, there are always cases where deviation from stated preferences may be justified. The most important rule is to provide a complete and clear description of part geometry.
General Rules and Preferences

1. In the United States, the American National Standards Institute (ANSI) drafting display format is preferred. This standard formatting controls all the display details such as the display of dimension text sizes, notes, line thicknesses, line formats, and symbol sizes. All dimension text and notes are to be horizontal. The CAD system will do this automatically when ANSI format is selected.

2. Draw and dimension the entire part. The use of the symmetry symbol to reduce the number of required dimensions is not a preferred practice.

3. Select a front view that best shows most of the important features of the part. When necessary, other views (top, bottom, right, left, and rear) are to be placed around the front view. The views must align and have the same scale.

4. Decimal millimeter values are used for dimensions in this book. When a value is less than one millimeter, a zero is to be shown before the decimal point. Trailing zeroes, however, are to be avoided.

5. Avoid odd dimensions or unusual precision unless well-justified. Most part dimensions can be adjusted slightly to make them more rounded. Using extra figures of accuracy where they are not needed would make the part more costly to make. A simple number rounding rule will be presented later in this section.

Figure 1-16 shows a part with three dimension specifications. Unusual and unnecessary precision should be avoided and the trailing zeroes should not be shown.

![Figure 1-16. Avoid unusual precision or trailing zeroes](image)
Dimension and Extension Lines

6. Do not use an existing line as a dimension line. This includes part profile visible lines, continuation of visible lines, hidden lines, centerlines, or extension lines. All dimension lines must be separately drawn.
7. Place dimension values centered between the arrows when there is sufficient space between the arrows.
8. Place dimension lines so they do not intersect.
9. Each dimension line must have its own end arrows. Arrows are not to be shared between dimensions.
10. Avoid using part profiles as extension lines.
11. When extending centerlines to be used as extension lines, use the centerline line format.
12. Minimize intersection between dimension lines and extension lines. Also minimize intersecting extension lines.
13. Point leaders to lines at an angle other than 0 or 90 degrees (preferably closer to 45 degrees).
14. Leaders pointing to circular features are to be radial in direction.

Figure 1-17 shows examples of inappropriate placement of dimension values and use of part profile lines as extension lines.

![Diagram](image)

**Figure 1-17.** Place dimensions outside the part and do not use part boundary as extension lines. Do not share arrows between two dimension lines.

15. Occasionally the part profile can be used as extension lines when separate extension lines become long and intersect many other lines creating confusion. A common example is in shafts with many features as shown in Figure 1-18.
Figure 1-18. Using part profile as extension lines is preferred for shafts with several features

**Dimension Placement**

16. Place dimension text outside the part view boundary or at least outside the part outer profile box. When space allows, place dimensions between views.
17. When dimension lines are too close, stagger dimension values so the text is easy to read.
18. Align the dimension lines when possible. Note that each dimension must have its own dimension line and its own arrows.
19. Maintain a uniform distance of about one character height between parallel dimension lines. The closest line to the part profile is placed at a distance of about 1.5 character height.
20. Keep the dimensions that define a feature close together and close to the feature and preferably on the same view. The reader should easily locate all of the dimensions related to a feature.
21. When it is necessary to put words on the drawings they should be all capital letters. However, standard symbols have been developed to minimize the use of words – use words only when there are no symbols to convey the message. For example, the word THRU is commonly used to indicate clearance holes or cuts when section views or hidden lines are not used to show the feature depth.
22. Favor feature-based dimensioning in which dimensions define size and location of features. Identify the features, dimension their internal geometry, and then use functional dimensions to locate the entire feature with respect to other features.
Repetition Symbol

23. Use the repetition symbol nX (such as 4X) for repeated features, preferably features that form a clear pattern and are countable on the same view. Do not use other text such as 4 PLCS or 4 HOLES, etc.
24. When features (referred to by nX) are not all visible on the same view, they should be clearly identifiable on other views. Common features for which nX is used are patterns of holes, bosses, slots, rails, fillets, and similar patterns.
25. Use nX when the referenced feature is clearly and uniquely identifiable, otherwise use separate dimensions.
26. Every feature count of nX must refer to a separate feature. The fillet that appears on the top and bottom of a shaft are the same feature and count as one.

Figure 1-19 shows a plate part with a pattern of four holes and a cross hole. Note that the use of THRU for the 6.6 mm cross holes is not appropriate because it is not clear whether the hole is through the first section of the plate or both. One may use THRU BOTH but the hidden lines are preferred here to indicate the extent of the cross hole. Also, the extended centerline on the side view indicates that the hole is centered relative to the thickness of the plate. The pattern of four holes is internally defined by 60 mm and 14 mm dimensions and then the pattern is located with respect to the left edges using 30 mm and 8 mm dimensions. The locating dimensions must be chosen to be functionally important. In this case the pattern location with respect the left face and bottom face are important.

Figure 1-19. Dimensioning of feature patterns
27. Place all the related dimensions of a feature close to the feature, close together, and preferably on a single view. A person reading a drawing should mentally construct a feature quickly with little effort.

28. Specify the important functional dimensions first. Designers that discuss possible modifications of the design would like to see the most important and functional dimensions on the drawing. For the remaining dimensions, specify them in a manner that is most convenient to manufacturing.

Figure 1-20 shows some of the other dimensions for the example part shown in Figure 1-14. When a dimension can be shown on two different views, select the view in which the dimension is easier to interpret. For example, the lower step height of 20 mm is easier to interpret on the front view than the side view.

![Figure 1-20. Complete dimensioning of the part](image)

As mentioned before, the appearance of centerlines in the side view indicates that the slot feature is centered to the 60-mm thickness feature. Without the centerlines the part is not clearly defined. By showing the 40 mm and 60 mm dimensions, the designer implies that these dimensions are functionally more important than the two 10 mm side dimensions.

29. Avoid over-dimensioning or repeated dimensions for the same feature. Over-dimensioning means specifying a dimension or angle that can be calculated from the already specified dimensions and angles. In cases where it is not clear that two features have the same dimension, repeated dimensions can be used for clarity.

30. Minimize the use of TYP which stands for “typical” unless the referenced features are obvious. A common application is for a part with a large number of fillets or rounds on its edges such as in molded parts.

31. Use section views to display internal features of a part, detailed views for small features, and cutout views for local hidden features.
Figure 1-21 shows a part with internal features best viewed as a cross-section view. Not all of the dimensions are shown on the figure.

32. Do not show tangent lines where a surface blends into another surface such as for fillets and rounds. An example is shown in Figure 1-22.
33. Use compact specification formats to efficiently dimension common features such as counterbored holes, countersunk holes, blind holes, chamfers, keyways, etc. In a compact format all of the feature dimensions are placed in the same spot.

The part shown in Figure 1-23 is a plate with a pattern of five countersunk holes, four chamfered edges, and a blind cross hole.
Figure 1-23. A plate with countersunk holes

The pattern geometry is defined by the diameter of the pattern circle and the number of holes and their angular separation. The pattern is located relative to the two side faces defining the width of the plate. The compact notation for holes is usually used for small holes and fastener holes.

34. Minimize the use of reference dimensions (dimensions enclosed in parentheses) which can be calculated from other existing dimensions. Reference dimensions are appropriate when they refer to identifying dimensions of standard stock or standard parts that are intended to remain as is.

35. Use dual dimensioning on features that have standard sizes in a different unit system than the one used on the drawing. Some companies require dual dimensioning for all dimensions.

In the channel drawing in Figure 1-24, reference dimensions (3 inch by 1.5 inch) are used as identifying dimensions to indicate the channel standard size – these values are not to be altered. These values are also provided in dual dimensions as inch units refer to the identifying size dimensions. Also note the compact dimensioning scheme of the slotted holes. Search the internet for the “dimensioning of slotted holes” to see plenty of examples. You can also search the internet for dimensioning of other common features.
36. When specifying threads, use the standard thread designations such as M6 x 1. Do not separately dimension the diameter of such threaded features.

37. Do not use process information to convey geometry information. For example, do not specify “USE 10-mm DRILL” in lieu of actual hole dimension limits. Also, do not use specifications that refer to standard parts like “CUT FOR #205 WOODRUFF KEY” as a substitute for specifying Woodruff keyway’s dimensions.

38. Show preference to dimensioning distances rather than angles. Angles are usually not shown unless the angle is functionally important.

39. Place feature dimensions on the views where the feature shape information is best seen. For example, when dimensioning a slot feature, dimension the view in which it is clear whether the feature is a slot or a rail. When dimensioning a cylindrical feature, place the dimensions on the view in which it is clear whether the feature is a hole or a boss.

40. Do not use center marks for fillets or arcs when no dimensions are necessary to locate their centers. Always show the center marks for cylindrical holes or bosses and show the axis of cylindrical features.

41. When a partial cylindrical feature is used as a fit feature (i.e. something fits into that feature), use diameter dimension – otherwise use radius dimension.

Some of these points are illustrated in Figure 1-25.
Note that the slots and the rail are dimensioned in views where their depth can be seen. The slot and rail depths, widths, and locations are shown close together. The holes are dimensioned in the section view where the feature type, size, and depth can be seen together. The geometry of the D-shaped boss is shown in the profile view where the D-shape is best seen. The center feature is located to the bottom edge (75 mm) and to the edge of the rail (40 mm).

**Pattern Dimensioning**

42. Use dimensions to fully describe the internal geometry of a pattern and then use dimensions to locate the pattern with respect to other functionally relevant features.  
43. Connecting the centerlines of holes in a pattern is preferred but they can be implied and left out when the pattern lines reduce the readability of the drawing.

The part shown in Figure 1-26 has a pattern of four counterbored holes. The pattern is 170 mm by 55 mm and it is located with respect to the lower left corner edges. The pattern connecting lines are not shown in this case. The shape of the slotted hole is defined by its length (120 mm) and its width (15 mm) while its location is specified by its distance to the left face (40 mm) and the lower face (40 mm).
You can think of a feature pattern as a complex piece of furniture. When locating a piece of furniture we pick two edges and specify the distance of these edges to the walls with two numbers. The rest of the dimensions are defined by internal dimensions. Always use a minimum number of dimensions to locate a pattern.

**Hidden Lines**

44. Do not show hidden lines unless they help show the extent of features not visible in other views. When required, only show the hidden lines for the necessary features.
45. Do not dimension to hidden lines unless it is clear and simplifies the dimensioning or when it reduces the number of views.

**Additional Examples**

Figure 1-27 shows the first dimensioning step for the shaft shown previously. The shaft has a number of features associated with power transmission application. The first step is to describe the shaft diameters, lengths, and functionally important shoulder-to-shoulder distances.
After the dimensioning of the shaft diameters, including the left threaded diameter, the other features need to be dimensioned. Each feature should be identified and completely described before going to another feature. The first two selected features are:

- The rectangular keyway on the right end of the shaft
- The shaft neck close to the rectangular keyway

Figure 1-28 shows the necessary views to dimension the rectangular keyway and the shaft neck. The width and depth of the keyway are described in the V-V section view while the length is shown in the top view. The neck detail is shown in a detailed view for clarity. To save space the detailed view and section view are not labeled.
Figure 1-28. Dimensioning of the rectangular keyway and the 2 mm wide neck

Note that the center plane of the keyway slot coincides with the axis of the shaft. This means the two features are aligned.

Two additional features of the shaft are:
- The cross hole for a spring pin
- The Woodruff keyway

Figure 1-29 shows two additional section views to dimension the cross hole and Woodruff keyway. The Woodruff key is a standard key with a 28 mm diameter, 6 mm thickness, and 11 mm height. The manufacturer calls for the width of the keyway to be 6 mm, the radius of the keyway to be 14 mm, and the depth of the keyway to be 7.5 mm below the surface. The 7.5 mm depth leads to a distance of 32.5 mm between the keyway bottom and the opposite face of the shaft. Search the internet for “dimensioning of Woodruff keyways” for some examples.

The size of the spring pin hole is indicated to be 5.1 mm to match the manufacturer’s recommendation for a 5 mm spring pin. The 20 mm functional location is specified on the section view. Finally, the compact chamfer notation should be added to describe the chamfer detail at the end of the shaft. The internal geometry of all the features is now defined and each feature is adequately located.
The part shown in Figure 1-30 is the drawing for a flange shape representing parts that attach to a shaft and usually have hubs, keyways, bearing holes, centering bosses, and fastener hole patterns. Flange-type parts are best shown using front and section views. Note that the inner D-shaped cylindrical hole is dimensioned as a diameter because this hole is intended to fit a shaft. Other cylindrical dimensions are shown in the section view where it is clear whether they are bosses or holes.
Missing dimensions and vague geometry information are serious errors and must be avoided. Always check for missing dimensions at the end by pretending to be a virtual machinist attempting to build all of the part’s features using a virtual milling machine and a virtual lathe. Going through this virtual building procedure catches missing or vague information and also alerts you about features that may be difficult to make economically. Subsequently the design may be altered for easier manufacturing.

Figure 1-31 shows the dimensioning method used to describe the geometry of the crank part. While dimensioning to the points of tangency between lines and curves is not preferred, in this case the 70 mm flat side distances are dimensioned to define the theoretical shape of the part.
1.7. **Figures of accuracy for dimensions and tolerances**

When a theoretical dimension need not be controlled with precision, it is a good practice and often most economical to round the dimension value up or down as much as possible without being objectionable for other reasons. For example, if the theoretical model for a holding bracket happens to have a precise dimension without a particular reason, consider dropping the unnecessary figures of precision. For example, if a bracket dimension, based on some strength calculations, comes out to be 247.652 mm, consider using 247, 250, or 240 mm instead. If the dimension is the result of some analysis other than those associated with precision fits, apply the 1% dimension rule. Simply calculate 1% of the dimension (in this case about 2.47 mm) and round the number up or down to within the 1% value. In this case, a rounded value can be any number between 245 and 250 mm.

If the theoretical dimension is a result of precision fit calculations, apply the 1% rule to the tolerance value to get a more rounded value for the theoretical dimension. For example, suppose the tolerance applied to a 247.652 mm size dimension is to be ± 0.8 mm. One percent of the tolerance is ± 0.008 mm. Add and subtract this 1% of the tolerance value to the theoretical dimension to obtain a high and a low acceptable value.
In this case the high value is 247.660 and the low value is 247.644 mm. Use the roundest number between these limits. In this case 247.65 mm is a good choice. The resulting specification becomes 247.65 ± 0.8 mm.

Exercise Problems

1. Use a CAD system to model and fully dimension the housing endcap shown in Figure 1-32. The outer diameter of the part is 52 mm. The other dimensions are up to you.

Figure 1-32. The front and back views of the housing endcap

2. Use a CAD system to model and fully dimension the turning support shown in Figure 1-33. The overall width of the part is 80 mm. The other dimensions are up to you.

Figure 1-33. The turning support
3. Use a CAD system to model and fully dimension the part shown in Figure 1-34. The shaft is 325 mm long. The 30 mm diameter cylinders fit to ball bearings. The keyway in the middle supports a gear and the keyway at the end of the shaft supports a belt pulley. There is a M8x1 threaded hole on one end of the shaft as shown in the figure. The drill hole for the threaded hole is 19 mm deep and the thread is 16 mm deep. The rest of the dimensions are up to you.

![Figure 1-34. A power transmission shaft](image)

4. Use a CAD system to model and fully dimension the part shown in Figure 1-35. The plate length is 200 mm and the counterbored holes are for 6 mm screws.
5. Find an actual machine part and measure its features. Then, use your CAD system to model and fully dimension the part. Include an actual picture of the part.

6. Search the internet for recommended dimensioning methods of other common features. Use your CAD system to model a simple part with a variety of common features and fully dimension the part.

Figure 1-35. A flange with counterbored holes
Chapter -2
A Design Engineer’s Overview of
Tolerance Statements - Part-I

2.1. Overview

The next two chapters present a designer-oriented overview of the GDT standard
tolerance specification statements. The objective here is to provide a basic
understanding of how a designer can call for geometric accuracy and how geometric
tolerancing statements are interpreted. Geometric tolerances along with dimensional
tolerances (± or dimension limits) comprise the designer’s toolbox for controlling the
geometric aspects of part features needed to ensure fit and function. This chapter
presents the tolerances of form and orientation. The following chapter continues with
tolerances of location and profile.

2.2. Interpretation of geometric tolerances

The interpretation of most geometric tolerancing statements is straightforward and
intuitive. Each tolerance statement can be interpreted by creating a theoretical gage
which serves as an acceptance template. The gage dictates how close the features should
be to their perfect geometry in order to be acceptable. The actual part is to be
conceptually held against the gage (or the gage can be held against the part) to check their
acceptability. Not only is the gage geometry important but the manner of holding the
actual part against the theoretical gage can be important as well. Tolerance control
statements disclose the information necessary to build the theoretical gages and define the
method of gaging. Theoretical gages reflect the true meaning of tolerance statements but
they may not be practical to build or use. Inspection personnel are trained to use
alternative procedures and come up with decisions that conform to those obtained by
theoretical gaging.

As an example, Figure 2-1 shows a theoretical gage and the theoretical gaging procedure.
The left side of the figure shows a theoretical gage made up of two planes and a small diameter cylinder that appears like a line. The right side of the figure shows the gaging process in which the actual part is aligned to the gage. This chapter and the next chapter present the details of how to define theoretical gages for each tolerance statement and how to align parts to such gages.

Most theoretical gages are three-dimensional. They are created from planes and cylinders and other 3D features. There are also tolerance statements that control feature cross-sections. These tolerance statements lead to two-dimensional gages or planar templates that are constructed on a plane.

When tolerance statements are applied to part surfaces or cross-sectional curves, the tolerance verification is straightforward and simply requires checking the subject feature surfaces or cross-sectional curves against a tolerance zone. When tolerance specifications are applied to axes or center planes, the tolerance interpretation must prescribe how the subject feature is to be derived from the surface features. The method of identifying the derived features is a part of the gaging process.

Theoretical gages along with the gaging instructions capture the true meaning of the tolerancing specifications. In describing a tolerance statement, designers only need to explain the gage geometry and the gaging procedure. Theoretical gages are easy to define as their geometry closely mimics the tolerance specification. Geometric tolerances allow design engineers to control a feature’s size, form, orientation, and location with respect to other features.

Figure 2-1. Theoretical gage (left) and theoretical gaging (right)
2.3. Dimensional tolerances

Dimensional tolerances refer to the direct dimension limits or ± tolerance values and can be applied to any dimension that defines a part. When dimension limits are applied to features known as features-of-size (FOS), the meaning of the dimension limits is explained in the GDT standard in terms of acceptance gages. Regular FOS are cylindrical, spherical, or width features (slots and rails) that are toleranced with dimension limits. When dimension limits are used to control size or location of other features, their meaning is not defined in terms of gages - they are defined in terms of measured dimensions. In other words, the inspector has to make a measurement that corresponds to the tolerated dimension and compare that against the limits. Since manufactured features have deformations and distortions, it is impossible to assign a unique measured value to such dimensions without subjectivity. For that reason, the use of dimension limits on features other than FOS is discouraged.

In this section, the meaning of dimension limits is explained when they apply to regular features-of-size. Precision fits usually involve regular FOS as these features are easier to manufacture with high accuracy. Dimension limits applied to regular FOS apply to all feature cross-sections. The theoretical gage for a cylindrical feature is made up of two circles on a plane – one at each limit of size as shown in Figure 2-2.

![Figure 2-2. Theoretical gages for checking the size limits of a pin](image)

The gaging procedure requires maneuvering the 2D gages to determine if the actual feature cross-sections can fit inside the larger circle and outside the smaller one. An inspector may use a caliper as an approximation of this gaging. The caliper can be set to 10 mm to define the smaller gage and to 12 mm to define the larger gage. The 12-mm caliper must clear all the cross-sections (GO gage) while the 10-mm gage should not clear any cross-section (NOGO gage). For width features, the gages are composed of pairs of parallel planes as shown in Figure 2-3.
In this case, the cross-sections are to be smaller than 12 mm and larger than 10 mm. A partial cylindrical surface, such as a shaft with a keyway, is also a regular FOS as long as the cross-section is defined enough such that a mating cylinder can fit in it without falling out. A cylindrical fillet, for example, is not a regular feature-of-size.

### 2.4. Geometric tolerances

Geometric tolerances are applied to features using a tolerance control box as shown in Figure 2-4. The feature that receives the specification is the *subject feature*. Note that in geometric tolerancing practice, the CAD model defines the exact, theoretical, or basic shape of a part and tolerance statements define the degree of deviation allowed from such a theoretical geometry. The deviations can be with regard to size, form, orientation, location, or any combination of these geometric aspects.
Table 2-1 shows the GDT standard tolerance types along with one example of their syntax.

<table>
<thead>
<tr>
<th>Sym</th>
<th>Specification</th>
<th>Example</th>
</tr>
</thead>
<tbody>
<tr>
<td>Form</td>
<td>Straightness</td>
<td>Ø 0.25</td>
</tr>
<tr>
<td>Flatness</td>
<td>0.25</td>
<td></td>
</tr>
<tr>
<td>Circularity</td>
<td>0.25</td>
<td></td>
</tr>
<tr>
<td>Cylindricity</td>
<td>0.25</td>
<td></td>
</tr>
<tr>
<td>Orientation</td>
<td>Angularity</td>
<td>0.25 A</td>
</tr>
<tr>
<td>Perpendicularity</td>
<td>Ø 0.25 M A</td>
<td></td>
</tr>
<tr>
<td>Parallelism</td>
<td>0.25 A</td>
<td></td>
</tr>
<tr>
<td>Location</td>
<td>Position</td>
<td>Ø 0.2 A B C</td>
</tr>
<tr>
<td>Concentricity</td>
<td>0.25 A</td>
<td></td>
</tr>
<tr>
<td>Symmetry</td>
<td>0.25 A</td>
<td></td>
</tr>
<tr>
<td>Circular Runout</td>
<td>0.25 A</td>
<td></td>
</tr>
<tr>
<td>Total Runout</td>
<td>0.25 A</td>
<td></td>
</tr>
<tr>
<td>Profile</td>
<td>Profile of a Line</td>
<td>Ø 0.2 A B</td>
</tr>
<tr>
<td>Profile of a Surface</td>
<td>Ø 0.2 A B</td>
<td></td>
</tr>
</tbody>
</table>

Table 2-1. Tolerance types and examples of their usage format

The following sections will present the meaning of the tolerances of form and orientation.
2.5. Tolerances of form

Tolerances of form limit the geometric deviations of feature form relative to their perfect or theoretical geometric form. Form is a geometric characteristic independent of size. The best distinction between form and size can be observed in the shape of sport balls such as a basketball. A basketball can be functional over a considerable range of sizes, but if it is not round enough, it would not bounce predictably and becomes useless. To be useful, a basketball requires much higher roundness or circularity precision than size precision. Table 2-2 shows the tolerances of form and the frequency of their use in fit-related applications by machine designers. A 5-star frequency rating means the usage is very high, and a 1-star rating means rare usage.

<table>
<thead>
<tr>
<th>Symbol</th>
<th>Specification</th>
<th>Typical Features</th>
<th>Frequency of Use</th>
</tr>
</thead>
<tbody>
<tr>
<td>⬛️</td>
<td>Flatness</td>
<td>Flat surfaces</td>
<td>*****</td>
</tr>
<tr>
<td></td>
<td></td>
<td>Center planes of slots or rails</td>
<td></td>
</tr>
<tr>
<td>⬜️</td>
<td>Straightness</td>
<td>Axis of cylinders</td>
<td>***</td>
</tr>
<tr>
<td></td>
<td></td>
<td>Straight edges</td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td>Straight cross-section lines</td>
<td></td>
</tr>
<tr>
<td>⬝️</td>
<td>Circularity</td>
<td>Circular cross-sections</td>
<td>**</td>
</tr>
<tr>
<td>⬝️</td>
<td>Cylindricity</td>
<td>Cylindrical surfaces</td>
<td>*</td>
</tr>
</tbody>
</table>

Table 2-2. Tolerances of form

Flatness and straightness are important fit-related specifications while circularity and cylindricity are needed for specific functional reasons such as sealing or lubrication-related applications.

2.6. Flatness

Flatness requires a planar surface, or the center plane of a width feature (a slot or rail), to be flat to within the specified tolerance value. An example of flatness specification is shown in Figure 2-5.
The theoretical gage for flatness is a pair of parallel planes defining an acceptance zone or a tolerance zone. The flatness tolerance value defines the separation between the tolerance zone planes. The theoretical gaging procedure for flatness requires maneuvering the gage (or the part) to determine if the entire surface of the feature can fit inside the tolerance zone of the gage. Figure 2-6 shows a flatness specification and its theoretical gage.
Note that flatness does not check parallelness and therefore a feature like the one shown in Figure 2-7 has acceptable flatness as it entirely fits within the gage.

![Feature with acceptable flatness](image)

**Figure 2-7.** A feature with acceptable flatness as the entire surface fits inside the gage

A common method for inspection of geometric tolerances is by using a coordinate measuring machine (CMM). A CMM can sample a surface at many points and use mathematical procedures (soft gaging) to simulate a theoretical gaging process. Using CMMs with soft gaging programs come closest to inspecting features according to the theoretical gages unless a high-precision physical gage can be made. Figure 2-8 shows the inspection probe of a coordinate measuring machine.

![Coordinate measuring machine probe](image)

**Figure 2-8.** A coordinate measuring machine probe
The stylus of the machine touches the feature surface at many points and sends the coordinates to the computer. For example, for inspecting the flatness of the surface in Figure 2-6, the machine can collect ten or fifteen coordinate points on the feature surface. Soft gaging is a mathematical procedure that can determine whether the collection of the points can fit inside a pair of parallel planes with a separation of 0.5 mm.

2.7. Meaning of symbol next to the tolerance value

Flatness control can also be applied to the center plane of width features. Figure 2-9 shows the meaning of the flatness control specification applied to a slot feature. The tolerance zone is as before, a pair of parallel planes with a separation defined by the tolerance value. The center plane points are identified by mid points of all intersecting lines extending between the two side faces of the slot. These intersecting lines are perpendicular to the sides of the widest parallel planes that fits inside the slot. The widest parallel planes are shown in red.

![Figure 2-9. The meaning of flatness control applied to a slot feature center plane](image)

For width features, the designer can also specify a symbol following the tolerance value. The symbol is the maximum material condition modifier and the symbol is the least material condition modifier. An example is shown in Figure 2-10.
Controlling the flatness of the center plane of a slot

The $\t$ symbol changes the subject feature from the center plane to the feature surfaces. The tolerance zone changes from the tiny zone of width 0.5 mm to an acceptance zone with a size of 119.5 mm. The size of the acceptance zone is obtained by subtracting the tolerance value of 0.5 mm from the smallest size of the feature which is 120 mm. Figure 2-11 shows the theoretical gage and an acceptable slot feature for the flatness specification of Figure 2-10.

If the slot center plane is a little less flat, or the slot is made a little smaller in size, the feature would not be acceptable. It is clear that the $\t$-modified specification applied to a slot controls the combined size and form such that the slot’s opening for fit would not be less than 119.5 mm. The $\odot$ modifier has a similarly defined meaning but since its
application to fit is limited, we will just focus on the \( \textcircled{M} \) modifier in this chapter. We will not see the use of an \( \textcircled{L} \) modifier until Chapter-10 on multipart fits.

Figure 2-12 shows an example of applying the \( \textcircled{M} \) modifier to a rail feature. When the \( \textcircled{M} \) modifier is applied to a rail feature, the gage has a slot shape. The size of the gage is calculated as the largest size of the rail (120.2 mm) plus the tolerance value (0.5 mm). The feature is accepted if it fits inside the slot.

![Figure 2-12. The specification and gage for the flatness of the center plane of a rail when an \( \textcircled{M} \) modifier is used](image)

The \( \textcircled{M} \) or \( \textcircled{L} \) symbols can be applied to other features-of-size especially cylindrical holes and bosses. The modifiers can also be applied to axis straightness tolerance, orientation control tolerances, and position tolerances as long as the features are features-of-size.

### 2.8. Straightness

Straightness requires the axis of a cylinder (or a surface of revolution) to be straight. Figure 2-13 shows an example of the axis straightness control on the top figure. The theoretical gage for this straightness statement, shown in the middle figure, is a cylinder
with a diameter equal to the tolerance value. The theoretical gaging procedure requires maneuvering the gage (or the part) to determine if the entire feature spine curve can fit inside the tolerance zone. The bottom figure shows a feature with a bent spine.

![Theoretical Gage](image)

**Figure 2-13. The theoretical gage for straightness and the spine of the actual feature**

The spine curve of a surface of revolution is defined by the center points of all the cross-sectional curves of the feature. Straightness can also be applied to surface elements that are theoretically straight.

When an \( \circ \) modifier is used following the 0.25 mm tolerance value, the gage becomes a tube of size 40.5 mm. The tube size is equal to the MMC size of the bar (40.25 mm) plus the stated tolerance of 0.25 mm at MMC. If the actual bar fits inside the gage tube, it is accepted. That means the combination of the bar size and bending leads to a bar that fits the gage and is acceptable.

**2.9. Circularity \( \circ \)**

Circularity control applies to cylindrical features or features that have circular cross-sections. This statement requires that specified cross-sections or all cross-sections of
selected features to be circular in shape to within the stated tolerance value. Figure 2-14 shows an example.

![Diagram](image)

**Figure 2-14. An example of circularity control statement**

The theoretical gage for circularity is a 2D (planar) annular zone between two concentric circles. The radial thickness of the annular zone is equal to the specified tolerance value. The size (diameter) of the gage is flexible. The circles can expand or shrink together as long as the radial thickness of the tolerance zone is unchanged. The gaging procedure requires cutting the feature with cutting planes to determine the cross-section curves and then maneuvering the gage to determine if every cross-section curve can fit inside a tolerance zone of a suitable size. The center mark for the gage circles are intentionally not shown to emphasize that the center of the gage need not align with any particular point during gaging. This means the gage only checks the circular form and not its location with respect to any other feature.

Note that the size tolerance of ±0.2 mm automatically enforces a circularity of 0.2 mm. The separate circularity specification is just for finer form control. Reducing the size tolerance to ±0.1 mm to achieve form control is wasteful as obtaining size precision is more difficult and costly than obtaining form precision.
2.10. Cylindricity

Cylindricity statement only applies to cylindrical features. This statement requires cylindrical surfaces to be cylinder-like in shape to within the specified tolerance value. The meaning of cylindricity is similar to circularity except that cylindricity is a three-dimensional tolerance that controls the form of the entire cylinder, while circularity is a two-dimensional tolerance that controls the form of circular cross-sections.

The theoretical gage for cylindricity is the zone between two coaxial cylinders. The radial thickness of the tolerance zone is equal to the specified tolerance value and is fixed. The diameter of the coaxial cylinders is flexible. The cylinders can expand or shrink together as long as the radial thickness of the tolerance zone is unchanged. The gaging procedure requires maneuvering the gage to determine if the entire cylindrical feature can fit inside a gage with a suitable size.

Figure 2-15 shows a cylindricity specification and a theoretical gage with a tolerance zone.

![Diagram of cylindricity gage]

Figure 2-15. An example of cylindricity control statement and its gage

The axis of the tolerance zone is intentionally not shown to emphasize that the gage only checks the form of the feature. Note that if the feature is made having a conical, barrel shape, or hour-glass shape, it would have circular cross-sections and would pass a
circularity specification but not the cylindricity specification. Form tolerances check the form of single features not their relationship to other features. For example, neither cylindricity nor circularity can assure that a subject feature is coaxial with respect to the axis of another feature.

2.11. Tolerances of orientation

Tolerances of orientation are used to control the orientation of individual features such as planar surfaces or axes of cylinders relative to other reference features. The reference features, often planar surfaces or axes of cylindrical features, are called datum features. The following is the list of orientation-control tolerances.

<table>
<thead>
<tr>
<th>Symbol</th>
<th>Specification</th>
<th>Typical Features</th>
<th>Frequency of Use</th>
</tr>
</thead>
<tbody>
<tr>
<td>///</td>
<td>Parallelism</td>
<td>Planar, Axis</td>
<td>***</td>
</tr>
<tr>
<td>⊥</td>
<td>Perpendicularity</td>
<td>Planar, Axis</td>
<td>*****</td>
</tr>
<tr>
<td>\</td>
<td>Angularity</td>
<td>Planar, Axis</td>
<td>**</td>
</tr>
</tbody>
</table>

To explain the meaning of orientation tolerances, it is necessary to add constraint geometry or datum feature simulators to the theoretical gages discussed so far. The datum feature simulators (abbreviated to datum simulators) are identified by the labeling of datum features in the tolerance statement. While labels can be any alphanumeric symbol the preferred practice is to use the alphabet symbols in sequence starting with the letter A. Good choice of labels that meets the expectation of engineers and production personnel improves the readability of the tolerancing scheme and reduces interpretation errors.

Datums can be any geometry that can limit a part’s translation or rotation. They mimic the way the part is placed into the assembly and constrained. For example, the datum simulator of a gage for a planar datum is a plane that constrains the translation and rotation of a part as shown in Figure 2-16. You can think of datum simulators as perfect physical surfaces that support a part just like mating part surfaces do when the part is placed into the assembly. Every datum label identified in the part drawing and referenced in a tolerance statement defines a datum simulator. Each datum simulator immobilizes some of the part’s degrees of freedom with respect to the gage.
Figure 2-16. An example of a datum label and a datum simulator

The datum simulator of a cylindrical hole datum is an expanding cylinder that fits inside the datum feature as shown in Figure 2-17.

Figure 2-17. An example of datum simulator for a hole datum feature

For most machine parts datum simulators consist of planes, expanding (contracting) cylinders, or expanding (contracting) width features that grip the part. If necessary, a planar datum simulator can be specified to translate to fully engage and constrain a part. Other datum simulators can be specified to be points, lines, or small areas that can support or hold a part. These datum simulators, called datum targets, are often used when part surfaces are uneven such as with cast parts. The primary application of the datum targets will be presented in Chapter-4 on default tolerances.
2.12. Parallelism  //

Parallelism requires a surface or axis to be parallel to a datum surface or axis to within the specified tolerance value. Figure 2-18 shows a table for which the top surface is specified to be parallel to the bottom surface with a tolerance value of 5 mm. The reference surface used for orientation control is identified as a datum feature, in this case datum-A.

![Figure 2-18. An example of parallelism control statement](image_url)

The theoretical gage for this parallelism specification is composed of a tolerance zone geometry for the subject feature and a datum simulator for the constraint geometry. The tolerance zone geometry is parallel to the datum simulator plane but otherwise floating parallel to the datum simulator. Figure 2-19 shows the gage for the specification shown in Figure 2-18.
The theoretical gaging procedure for this parallelism specification requires two steps. The first step is for the part’s datum-A feature to be firmly pressed against the datum simulator (Plane-A in the figure). Aligning the part to gage’s datum simulators is called staging. Staging is the procedure that constrains the part with respect to the gage before the subject feature is checked against the tolerance zone geometry. The second step is to slide the tolerance zone parallel to the datum simulator to determine if the entire subject feature can be made to fit inside the tolerance zone. The gaging procedure is shown in Figure 2-20. In this case, the table-top is not sufficiently parallel to the bottom support because regardless of how the tolerance zone is moved, the entire feature cannot fit inside the zone.
Note that the size tolerance of 20 mm controls the parallelism to within 20 mm. If the size specification is changed to 700 – 705 mm, the desired level of paralleness is achieved. However, controlling parallelism or other orientation precision needs through size control is wasteful as the needed size control is more difficult and costly to achieve than orientation control.

Tolerances of orientation or location frequently involve cylindrical datum features or cylindrical subject features. The following section explains how the axes of cylindrical features are established.

2.13. Identifying the axis of an imperfect cylinder

Consider a parallelism statement that requires the axis of a cylindrical subject feature to be parallel to the axis of another cylinder chosen as a datum feature. An example is shown in Figure 2-21.

![Figure 2-21. A cylindrical feature to be parallel to another cylindrical feature](image)

When the datum feature is a cylindrical hole, the simulated datum is an expanding pin. During gaging, this pin fits inside the datum hole and expands until it fits fully inside the hole as shown in Figure 2-22.
Figure 2-22. Identifying the axis of a cylindrical feature

There is a special name for the largest cylinder that tightly fits inside a hole – it is called the unconstrained actual mating envelope (unconstrained AME). The term unconstrained refers to the fact that the expanding pin is allowed to freely center itself to assume the largest size. The actual mating envelope and the actual feature behave the same in a fit situation but the AME is perfect and its axis is meaningful. When showing the theoretical gages, an expanding cylinder datum simulator will be shown as an axis with a small circle in dashed lines.

The tolerance zone geometry of the gage is a long cylindrical zone parallel to the datum simulator axis with a size equal to the tolerance value. For example, Figure 2-23 shows the theoretical gage for the specification shown in Figure 2-21. Axis-A, shown in the figure, is the axis of an expanding cylinder that fits into the datum cylinder during gaging. The dashed circle around the axis is an indicator that the axis is established by an expanding (or contracting) cylinder.

Figure 2-23. Theoretical gage for parallelism of a hole with respect to another hole
For the part staging, the part is constrained by an expanding pin (datum simulator) that fits inside the datum feature hole. Then, the axis of the subject feature’s unconstrained AME is identified. This subject feature axis is checked for containment into the tolerance zone cylinder. The tolerance zone cylinder is free to translate parallel to Axis-A to the best location for axis containment check.

Figure 2-24 shows the unconstrained AME for a hole feature and also for a shaft feature. The unconstrained AME for a shaft is the smallest hole that fits over the shaft.

![Diagram showing unconstrained AMEs for a hole feature and a shaft feature.](image)

**Figure 2-24.** Identifying the axis of a cylindrical feature using unconstrained AMEs

The concept of the actual mating envelope also applies to slot and rail features. In the case of slots and rails, the expanding or contracting cylinders are replaced by expanding or contracting parallel planes or jaws. Figure 2-25 shows the unconstrained AME for a slot feature.
The unconstrained AME establishes the center plane of slot or rail features. As you may have guessed, there are also constrained AMEs which are only applicable to datum features when the tolerance statement includes multiple datum specifications. Constrained AMEs simply mimic the way the part is placed into an assembly. The derived axis, or center plane of subject features, however, will always be the unconstrained AME. Constrained AMEs will be discussed when position tolerances are introduced in the next chapter.

When the tolerance value is followed by the \( M \) modifier for a hole, the tolerance zone (acceptance zone) of the gage becomes a bar. Such a tolerance statement controls the combined effect of feature size and the deviation of its axis from perfect parallelness with respect to the datum axis. An example of a \( M \) modified specification and the resulting theoretical gage is shown in Figure 2-26.
After aligning the datum simulator axis of the gage with the actual datum feature axis, the hole feature must fit over the gage acceptance zone (pin). The acceptance zone is free to translate parallel to the datum axis. The size of the pin for a hole feature is 39.7 mm which is equal to the smallest hole size (40 mm) minus the tolerance value (0.3 mm). When the actual hole is made to be larger (closer to 40.5 mm), the gage automatically allows it to be less parallel relative to the datum axis. When the pin is closer to 40 mm its axis has to be more parallel relative to the datum axis.

2.14. Perpendicularity

Perpendicularity requires a surface or axis to be perpendicular to another reference surface or axis to within a specified tolerance value. The reference surface or axis is labeled as a datum. Figure 2-27 shows an example in which an axis is specified to be perpendicular to a planar feature.
The interpretation of perpendicularity tolerance statement is similar to that of parallelism. The only difference is that the tolerance zone of the gage is perpendicular to the datum simulator plane or axis. The theoretical gage for the perpendicularity statement of Figure 2-27 is shown in Figure 2-28.
The gaging process requires placing the datum feature of the part against the datum simulator and checking the axis of the boss feature for containment within the 0.3 mm diameter tolerance zone. The axis of the boss subject feature is established by its unconstrained AME.

If an \( \circ \) modifier follows the tolerance value in Figure 2-27, the acceptance zone of the gage becomes a hole of size 30.35 mm. The gage controls the combined feature size and orientation. Figure 2-29 shows the resulting gage and an example of gaging. The outline of an actual part is shown in red. This feature fits the acceptance zone of the gage while fully resting on the datum simulator.
Figure 2-29. The theoretical gage for axis perpendicularity statement when tolerance value is followed by an @ symbol

Note that the gage allows a pin made closer to its LMC size of 30 mm to have a larger orientation deviation than a pin closer to its MMC size.

2.15. Angularity

Angularity requires a surface or axis to be at a certain angular orientation relative to another reference surface or axis to within a specified tolerance value. The reference is identified as a datum. Figure 2-30 shows a typical use of angularity for a planar feature.
Figure 2-30. An example of angularity tolerance statement

The gage for this angularity tolerance specification is shown in Figure 2-31.

Figure 2-31. Theoretical gage for the angularity tolerance

The interpretation of angularity follows the same procedure as parallelism and perpendicularity. For the angularity statement shown in the figure, the tolerance zone is a
pair of parallel planes with a 0.1 mm separation. This tolerance zone is oriented at an angle of 28 degrees relative to the datum simulator plane of the gage.

**Exercise Problems**

1. Model the turning support part in a CAD system and create a drawing sheet as shown in Figure 2-32, excluding the labels.

![Figure 2-32. Turning support part](image)

Using GDT standard annotation capacities of the CAD system, specify the following tolerances:

a. Define plane P₁ as Datum-A.

b. Define slot S₁ as Datum-B.

c. Define plane P₂ as Datum-C.

d. Make Datum-A feature flat to within 0.02 mm.

e. Make Datum-B perpendicular to Datum-A to within 0.05 mm.

f. Make Datum-C perpendicular to Datum-A and Datum-B to within 0.1 mm.

g. Specify the size limits of hole H₁ to be 20 ±0.02 mm.

h. Specify the size of slot S₁ to be 76.00 – 76.15 mm.

i. Specify the size of holes H₂ to be 12.00 – 12.25 mm.

j. Specify hole H₁ to be parallel to plane P₁. Use a tolerance value of 0.1 mm.
2. Consider the turning support part shown in Figure 2-33. Draw the theoretical gage for the following:
   a. The flatness specification.
   b. The circularity specification.
   c. The parallelism specification.
   d. The perpendicularity specification.

3. Consider the turning support part shown in Figure 2-34. Draw the theoretical gage for the following:
   a. The straightness specification.
   b. The parallelism specification.
   c. The perpendicularity specification.
   d. The cylindricity specification.
4. Consider the part shown in Figure 2-35.

The manufactured part is shown in Figure 2-36.
   a. Create theoretical gages for the two tolerance statements.
b. Draw the perpendicularity gage and the center plane of the slot feature on the manufactured part such that conformance, or lack of conformance, is shown. Is the orientation of the slot acceptable?
c. Draw the perpendicularity gage for the planar feature on the manufactured part such that conformance, or lack of conformance, is shown. Is the orientation of the planar feature in the perpendicularity statement acceptable?

Figure 2-36. A 2D manufactured part

5. Consider the specifications shown in Figure 2-37.
   a. Draw the gage on the manufactured part such that conformance, or lack of conformance, is shown for the perpendicularity statement.
   b. Is the perpendicularity of the plane acceptable?
   c. Check the circularity of the hole feature and draw the circularity gage on the part.

Figure 2-37. A 2D part drawing
Figure 2-38. A 2D manufactured part
Chapter-3
A Design Engineer’s Overview of Tolerance Statements - Part-II

3.1. Overview

This chapter presents the meaning of the remaining important tolerance statements including tolerances of location and profile tolerances. A gage that defines a tolerance of location must use at least one location dimension for its construction. Sometimes the location dimension is zero when the features are aligned such as with coaxial features. The location dimension can also be the distances within a pattern of features.

3.2. Tolerances of location

Tolerances of location control the distance between a subject feature and one or more datum features. Tolerances of location fall into one of the types shown in Table 3-1.

<table>
<thead>
<tr>
<th>Symbol</th>
<th>Specification</th>
<th>Typical Features</th>
<th>Frequency of Use</th>
</tr>
</thead>
</table>
| ✅     | Position tolerance | Single or pattern of cylinders  
              Single or pattern of width features | ***** |
| ⚔️     | Circular runout | Circular cross-sections (2D) | ***** |
| ⚔️     | Total runout | Cylindrical features | *** |
| ⚔️     | Concentricity | Circular cross-sections (2D) | * |
| ⚔️     | Symmetry | Cross-sections (2D) | * |

Table 3-1. Tolerances of location

3.3. Position tolerance ✅

Position tolerance may be applied to features-of-size or their patterns. A pattern of features must have features with identical sizes and usually with identical functions. For single features, position tolerance requires the axis (or center plane) of a subject feature to be within a specified tolerance zone with respect to a datum frame. For patterns of features, the position statement additionally requires the relative locations within the pattern to be maintained with the same accuracy. The presentation here uses cylindrical subject features but the position tolerance also applies to width features. For width features, center planes substitute for the axes of cylindrical features.

Figure 3-1 shows a simple part with a position control statement. Assume the parts shown are plates with a constant thickness.
The theoretical gage for this position tolerance specification is shown in Figure 3-2. Datum-A and datum-B are datum feature simulators.

Since the position tolerance applies only to features-of-size the tolerance value can always be followed by an \( M \) or \( L \) modifier. The position tolerance statement with
modifiers are used to control the combined effect of size and position deviation. Figure 3-3 shows an example of an $\mathbb{M}$-modified tolerance value.

![Figure 3-3. An example of an $\mathbb{M}$ modified position tolerance specification](image)

For an $\mathbb{M}$-modified tolerance statement, the acceptance zone of the gage becomes a boss as shown in Figure 3-4. The size of the boss is equal to the minimum size of the hole (15 mm) minus the tolerance value (1 mm) resulting in a 14 mm boss. The gage allows a hole closer to its LMC size of 16 mm to deviate from the theoretical position more than a hole made closer to its MMC size 15 mm.
When multiple datum features are used in a tolerance statement, datum references are placed in the tolerance control box and their order or priority is assigned from left to right. The datum priority makes a difference in the manner of gaging but the gage itself is unaffected by the datum ordering. Up to three datums can be referenced in a position tolerance statement. The first datum is called the primary datum, the next one, if specified, is the secondary datum, and the third one, if specified is the tertiary datum.

The datum features and datum priority mimic the way the part is intended to be fixed to an assembly. Figure 3-5 shows a final desired arrangement of a part into an assembly.
Each datum reference in a tolerance statement leads to a corresponding datum simulator on the theoretical gage. These datum simulators represent the perfect-form and perfect-orientation mating surfaces. The procedure for constraining the part to the gage is stated through the order of datum specification and illustrated in Figure 3-6.

The primary datum simulator makes full unconstrained contact with its datum feature on the part. The secondary datum contact would not have an unconstrained contact because the part is required to remain in contact with the primary datum simulator. If the manufactured part had perfectly perpendicular datum features, then the datum priority would have made no difference when placing the part into assembly.

Figure 3-7 shows the actual arrangement of the part relative to the datum simulators when the datum priority is reversed. This change of datum priority leads to a completely different part arrangement. In this case, datum-B surface orients the part.
Once the part is constrained to the datum simulators, the subject feature axis can be checked against the tolerance zone of the theoretical gage as shown in Figure 3-8. The basic position is the theoretical position of the hole axis with respect to the datum frame. The basic position is where the axis should be for a perfect placement.
In the case of Figure 3-8, the hole position is out of tolerance. Since simulators represent the mating surfaces on which the part contacts, the selection of the datum priority is such that it mimics the final assembly. When a part is being installed into an assembly it can only make full unconstrained contact on one surface only. The next contact is going to be partially constrained by the first primary contact. For example, if datum-A plane is fixed to its mating plane by screws, then datum-A plane would be the primary datum. If datum-B plane is screwed to its mating part when it is installed in the assembly, then datum-B plane should be specified as the primary datum. If the design requires both surfaces to be screwed to mating surfaces, then the datum surfaces must be specified to be perpendicular with high precision. In that case it makes no difference which datum is selected to be the primary datum.

Figure 3-9 shows another example of a position tolerance involving a bearing hole controlled with respect to two datum references. The 25H7 designation is a coded way of stating the size limits of the hole. The 25H7 numerical limits are 25.000 – 25.021 mm. Coded size limits will be presented later in Chapter-6.

Figure 3-9. An example of position tolerance statement

In this example, datum-A is the primary datum. Datum-B, the secondary datum, is the center plane of the slot. To specify the center plane as the datum, the datum symbol must be aligned and attached to the dimension line. The axis of the bearing hole is
required to be parallel to and at a precise distance from datum-A. The bearing hole is also required to be centered relative to the slot center plane identified as datum-B. The datum simulator for datum-A is a plane and the datum simulator for datum-B is a pair of expanding jaws that grip the part. The expanding pair of jaws, however, must remain perpendicular to the primary datum-A simulator as the jaws open. Rather than showing the expanding pair of jaw datum simulators, the center plane of the expanding jaws will be shown in the theoretical gages. The theoretical gage for this statement is shown in Figure 3-10.

![Figure 3-10. The theoretical gage](image)

In the theoretical gaging procedure, which is shown in Figure 3-11, first the bottom surface of the part is pressed against the datum-A simulator of the gage just like a part is placed into a mating part surface of an assembly. Next, the parallel jaws of datum-B simulator grip the part as they open to fit inside the slot datum feature. The datum-B simulator mimics the slot fitting tightly into a mating rail.
Figure 3-11. Theoretical gaging procedure requires constraining the part to the datum planes and checking the subject feature axis for containment within the tolerance zone.

Figure 3-12 shows a pair of expanding jaws (parallel planes) gripping the slot. The resulting boundary (when the jaws can no longer expand) is the orientation-constrained actual mating envelope (orientation-constrained AME) of the slot.

Figure 3-12. The orientation-constrained actual mating envelope for a slot
After the part is aligned to datum simulators, the axis of the bearing hole is checked for containment against the tolerance zone cylinder. The axis of the bearing hole (subject feature) is always established by the axis of its unconstrained AME (the largest expanding cylinder fitting inside the bearing hole). If the tolerance is modified, the hole must clear an acceptance bar of size 24.92 mm. The bar size is equal to the MMC size of the hole (25 mm) minus the tolerance value (0.08 mm).

Position tolerance is frequently used to control the position of a pattern of holes, or center planes of slots, with respect to datum features. The position tolerance also controls the relative locations of the features within the pattern. Figure 3-13 shows a pattern of four holes and a position tolerance statement applied to the pattern.

The pattern of four holes is to be primarily perpendicular to the bottom surface designated as datum-A. A primary datum plane is commonly called the “primary orienting datum” although it also locates features as well. When starting to tolerance a part for fit, the first question you should ask is “what surface orients this part with respect to the mating part or assembly?” Do not underestimate the importance of the primary orienting datum. The pattern must also be precisely located with respect to the top plane designated as datum-B. A secondary datum surface is commonly called the “secondary locating datum”. Finally, the pattern must be precisely centered with respect to the center.
plane of the rail feature designated as datum-C sometimes called the “tertiary locating datum”. Figure 3-14 shows the resulting theoretical gage. Do not be alarmed by the zero tolerance value in the perpendicularity tolerance. A zero geometric tolerance value is not an error when used with an \( \mathcal{C} \) modifier.

![Diagram](image)

**Figure 3-14. The theoretical gage for inspecting the pattern position tolerance statement**

The gage is composed of three mutually perpendicular planes. Datums-A and datum-B are contacting datum simulators while datum-C represents the center plane of collapsing jaws that grip the datum-C rail. The pattern of four tolerance cylinders has a theoretical distance relationship with datum-B and datum-C planes. Tolerance cylinders have 0.5 mm diameters. Figure 3-15 shows how the gage is aligned to the part to check the position accuracy of the hole pattern. The datum simulator for the rail feature, datum-C, is a pair of collapsing jaws that must remain perpendicular to datum simulator plane A and datum simulator plane B. The theoretical gage shows the center plane of the jaws after they fully grip the part.
Figure 3-15. The theoretical gaging for inspecting the pattern position tolerance statement

In this example the datum frame completely fixes the part to the gage. Judging from Figure 3-15, two of the holes in the pattern are clearly off position. Figure 3-16 shows only the datum simulators of a physical gage.

Figure 3-16. A physical gage showing the three datum simulators
If an $\Theta$ modifier follows the tolerance value as indicated in Figure 3-17, the tolerance zones change to a pattern of acceptance bars. The $\Theta$ modified specification controls the combined effect of size and position deviations. The gage automatically allows larger holes to be off position more than the smaller holes.

![Diagram showing position tolerance controlling a pattern of holes using $\Theta$ modifier](image)

Figure 3-17. Position tolerance controlling a pattern of holes using $\Theta$ modifier

The resulting theoretical gage is shown in Figure 3-18. The gage appears more realistic now as a mating part than the gage with tiny laser beam tolerance zones. After all there is no physical line in the middle of the hole pattern to fit anything. The size of each bar is 9.25 mm.
Figure 3-18. The theoretical gage for the modified pattern position tolerance

Figure 3-19 shows the theoretical gage and the gaging process. The hole features are acceptable if they all clear the gage bars.

Figure 3-19. The theoretical gaging for inspecting the pattern position tolerance statement

A physical gage simulating the theoretical gage and the gaging procedure can be constructed as shown in Figure 3-20. The loose pins are used to make the physical gaging practical.
After the part is placed on the datum-A simulator plane and pushed against datum-B, the jaws move to grip the part to complete datum alignment. As the jaws grip the part the part must remain in full contact with datum-A and secondary contact with datum-B surfaces. The inspector then attempts to insert the pins into their sockets that have the same sizes as the pins. If all the pins fit into their sockets, the pattern geometry is acceptable regarding the position tolerance. The jaws must be designed such that they move together the same amount. This ensures the hole pattern will always remain centered to the jaw’s center plane.

3.4. Meaning of \( @ \) modifier applied to datum features

It was mentioned that datums simply mimic the constraining surfaces of an assembly as the part is placed into the assembly. It would be a very odd design for a machine part to have expanding jaws or shrinking cylinders that collapse on bars or rails. Therefore, center plane or axis datums do not reflect the way datums work in fit-related applications. When a datum feature referenced in a tolerance statement is a feature-of-size and intended to fit a mating feature, it is most likely a rigid feature. The designer can apply a material condition modifier to such a datum. When an \( \ominus \) modifier is used, the datum is said to apply at its maximum material boundary (MMB). An example is shown in Figure 3-21.
A referenced datum feature with an \( M \) material condition modifier establishes a fixed size datum simulator. The size of the datum simulator is determined using the same formula as the size of \( M \) modified subject feature acceptance zones were determined. In this case, the size of the datum simulator slot for datum-C is 30.15 mm. The size of datum-C is obtained by adding the rail’s MMC size of 130.15 mm to its tolerance of zero at MMC. Figure 3-22 shows the physical gage for the inspection of the part with MMB specification for datum-C. As you can see this is a more realistic representation of a mating part compared to one with the expanding jaws.
The purpose of applying an \( \oplus \) modifier to a datum feature is to allow any slack in the datum fit to compensate for inaccuracies of the subject feature pattern during assembly by adjusting the datum fit. In this case, any slack in the fit between the actual datum-C feature and the gage can be used to compensate for the feature pattern shift provided that such an adjustment can be made during assembly.

### 3.5. Composite position tolerance – multiple segments

In the example previously shown in Figure 3-13, a single position statement controlled the location and orientation of holes within the pattern and with respect to datums A, B, and C. After the part is fixed to a mating part using the stated datum frame, the designer may want to apply additional precision requirements to the same pattern of features. Imagine you are asked to move a dining table into a room. The room’s floor and walls play the role of reference or datum surfaces. The table’s four feet act as a pattern of features for locating and orienting the table. A position tolerance is like placing four large circular patches on the floor for the table feet. The patch centers define the theoretical distance between the table feet and their theoretical distance to the walls. As long as every foot is within a marked patch the table position is acceptable.

To control the position and orientation of a pattern more precisely with respect to only some of the datums in the datum frame, another position control statement can be added.
to the original specification. This additional position tolerance, which has a smaller tolerance value, also controls the relative position of the holes with respect to each other with more precision. This type of position tolerancing is called multiple segment composite position tolerance and only applies to feature patterns. In the dining table analogy, there will be four additional smaller circular patches placed at the theoretical distance to one of the walls but able to slide relative to the other wall.

Figure 3-23 shows a multi-segment composite position tolerance. Note that there is a separate position tolerance symbol for each statement. The upper segment controls the pattern position with respect to the three datum planes as before. The lower segment controls the position of the pattern more precisely with respect to two datums, datum-A and datum-B. It also controls the relative location of the features within the pattern with more accuracy.

Figure 3-24 shows the gage for the composite tolerance. The gage now includes the 0.25 mm tolerance zone pattern (shown in green) that is not centered with respect to the rail (datum-C) and therefore is free to slide while maintaining its basic distance to datum-B.
Figure 3-24. The theoretical gage for multiple segment composite position tolerance statement

Figure 3-25 shows the gage for the lower statement. The designer controls the pattern location more closely with respect to datum-A and datum-B without needing the additional level of precision with respect to datum-C. The circles represent the tolerance zones. The arrows show that the tolerance zones can translate as a whole to the right and left (parallel to datum-B) but not up and down (perpendicular datum-B).

Figure 3-25. The theoretical gage for inspecting the pattern position with respect to datum-A and datum-B
If the tolerance values are \( M \) modified, the tolerance zones will become acceptance bars of size 9.5 mm for the upper position statement. The specification then controls the combined effect of size and position deviation. For an \( R \) modified lower specification, the 0.25 mm tolerance zones become 9.5 mm acceptance bars.

If the two statements with the composite format are applied independently as two unconnected statements, then each one would have resulted in a separate gage. The difference is that in the composite format the datum simulators are aligned with the part once and the features are checked against the tolerance zones using the same alignment. In other words, the staging is the same for all composite tolerance statements. The practical concept behind multiple segment tolerances is that once the part is placed and fixed into an assembly using its datums, the pattern is to meet other accuracy requirements without adjusting the datum contacts.

A theoretical inspector first aligns the datum frame with the part datum features and then checks the red tolerance zones associated with the upper tolerance segment. If the pattern passes this stage, the inspector discards the red tolerance zones and slides in the green lower segment tolerance zones to check the lower segment. If the pattern passes the upper tolerance statement it means that:

1. The pattern has internal location precision at 0.5 mm level. The pattern is also located to datum-B and centered to datum-C at 0.5 mm precision level.

If the pattern also passes the lower tolerance statement it means that:

2. The pattern has high internal location precision (all features are located and oriented with respect to each other with high precision (at 0.25 mm level).
3. The pattern is square to datum-A with high precision (at 0.25 mm level).
4. The pattern is located with respect to datum-B with high precision (at 0.25 mm level).

Note that in order for the two attached position tolerance statements to be considered a composite tolerance, the datum frame must be repeated exactly in the lower segment, including its order, but with fewer datum constraints. If the datum frame does not repeat exactly in the lower segment, the two statements are interpreted as two independent position tolerance statements.

If the designer intends to make the pattern position even more precise in terms of being square to datum-A and also for the relative positions within the pattern, a third statement can be added to the composite position statement using only datum-A reference. This leads to a gage with a third pattern of tolerance zones with even smaller tolerance zone cylinders. Three-segment composite tolerance statements, however, are not common.
3.6. Composite position tolerance – single segment

A composite position tolerance can be specified in which a single position tolerance symbol is shared by multiple statements. This is called single-segment composite tolerance format. An example is shown in Figure 3-26.

![Figure 3-26. A single-segment composite position tolerance statement](image)

In a single-segment position tolerance format, the upper statement is interpreted as in the multi-segment format. The upper statement locates the pattern with respect to the datum features as before. The lower segment, on the other hand, only controls the orientation of the pattern with respect to the specified datum frame as well as the relative location of the feature pattern members. In the dining table example, the table feet would be required to remain more parallel to the walls within their allowed larger position circles. Figure 3-27 shows the tolerance zones resulting from the lower statement of the single-segment tolerance specification. The part hole pattern need not be located to datum-B anymore as long as it remains parallel to datum-B. The upper segment still controls how far the pattern can translate toward or away from datum-B.
Figure 3-27. The theoretical gage for inspecting the pattern position with respect to datum-A and datum-B in a single-segment specification

For example, the lower segment in Figure 3-26 controls the orientation of the pattern more precisely with respect to datum-A and datum-B. The lower statement also controls the relative location of the pattern features with more accuracy relative to one another. The result of the composite statement is that the pattern is allowed to translate toward or away from datum-B more but it is not allowed to rotate too much. The theoretical gage for this statement is identical to the one for multiple-segment position tolerance format except that the tolerance zone associated with the lower segment is not located and is free to translate away or toward datum-B. If the pattern passes the upper tolerance statement it means that:

1. The pattern has internal location precision at 0.5 mm level. The pattern is also located to datum-B and centered to datum-C at 0.5 mm precision level.

If the pattern also passes the lower tolerance statement it means that:

2. The pattern has high internal location precision (all features are located and oriented with respect to each other with high precision (at 0.25 mm level).
3. The pattern is square to datum-A with high precision (at 0.25 mm level).
4. The pattern is parallel to datum-B with high precision (at 0.25 mm level).

In order for the two position tolerance statements to be considered a composite tolerance, the datum frame must be repeated exactly in the lower segment, including its order and modifiers. If a designer intends to make the relative locations within the pattern even more precise and more square to datum-A, a third statement can be added to the composite position statement using only datum-A reference. This leads to a gage with a third pattern of tolerance zones having smaller size tolerance cylinders. This type of three-segment composite tolerance statements is also rarely used.
3.7. Runout tolerances

Circular Runout

Circular runout requires feature cross-sections to be circular in form and concentric to the axis of a feature selected as datum, to within the specified tolerance value. Circular runout is primarily used to control the coaxiality of shaft cross-sections relative to an axis of rotation. Figure 3-28 shows the shaft assembly of a gearbox. In this application, it is important for the gear’s fitting surface to be coaxial to the axis of rotation defined by the two bearings. Otherwise, the gear teeth will wobble radially in and out shortening the life of the gear.

Figure 3-28. A rotating part that requires coaxiality of its features with respect to the axis of rotation defined by the bearings

Figure 3-29 shows how this type of tolerance control can be specified using circular runout specification. The 10k6 and 10h9 codes define the feature size limits. The application of these size codes will be covered in Chapter-6.
Figure 3-29. An example of coaxiality control through circular runout statement

Datum-A defines the axis of the smaller bearing fit cylinder. The larger bearing fit cylinder is controlled to be coaxial to datum-A with high precision using circular runout specification. The axis of the larger bearing cylinder is identified as datum-B. The gear’s fit surface is specified to be primarily coaxial to an axis defined by the two bearing axes simultaneously. The lack of circularity also influences circular runout.

The primary datum in circular runout specification is always an axis obtained by a collapsing (or expanding) datum simulator. The tolerance zone is the 2D zone defined by a pair of concentric circles with a radial thickness equal to the specified tolerance as shown in Figure 3-30. The tolerance zone is similar to circularity zone but there is an added datum point in the center of the circles.
The theoretical inspection requires intersecting the gear surface with a cutting plane perpendicular to the datum axis. The result is an intersection profile representing the feature cross-section, and a point representing the intersection of datum axis and the cutting plane. This is shown in Figure 3-31.

In this case, the cross-section is circular enough but it is off center with respect to the datum axis location. If we move the gage and align its center point with the datum intersection point, the cross-section profile would not fit inside the gage tolerance zone at any size.Circular runout specification is easy to verify using a simple inspection set up such as the one shown in Figure 3-32.
To do the physical inspection, the part ends are fixed in rotating chucks (collapsing cylinders). These chucks locate the center of rotation and establish the datum axis. The chucks are simultaneously tightened so neither of the datum cylinders is favored. The plane of the dial indicator simulates the cutting plane. The dial stem is perpendicular to the axis of the chuck and radially aligned with it. The part is then rotated with the dial indicator set up to measure radial deviations. As the part rotates, the full movement of the indicator (FIM), or the range of the dial travel, measures the runout of the cross-section relative to the axis of rotation. If this measured runout is smaller than the tolerance value, the measured cross-section is within tolerance. For checking another cross-section, the dial is moved to that cross-section and the process is repeated. Unless otherwise indicated on the drawing as a note adjacent to the runout specification, the runout applies to all cross-sections of the feature.

Circular runout can also be used to control the perpendicularity of shaft shoulders with respect to the axis of rotation. This is the specification of choice when controlling shaft shoulders that support bearings, gears, or other shaft components. Figure 3-33 shows an example.
Circular runout specification applied to planar surfaces is also easy to verify using a simple inspection set up such as the one shown in the top of Figure 3-34.
Unless otherwise specified, the specification applies independently to all circles on the subject plane. That means the dial indicator test must be applied at various circle sizes. When the surface bulges out or in, as shown in the part in the middle of the figure, no runout would register on the dial. The function of a bearing or a similar component that is fitted to a rotating shaft is not adversely affected by this kind of surface. The part in the bottom of the figure is unacceptable and, when tested, shows a lot of runout. This kind of surface forces the bearing or the component to rotate relative to the axis of rotation.

If you are curious to know, the theoretical gage for the specification in Figure 3-33 is a strip of cylindrical surface and a datum axis as shown in Figure 3-35. The axial length of the strip is the tolerance value. The strip is free to translate along the datum-A axis. The diameter of the cylinder corresponds to the location along the shoulder where the runout is checked.
Theoretical Gage

0.1

Datum-A

To check the part against this gage, the axis labeled datum-A is aligned with the actual A-B datum of the part. Then, the cylindrical tolerance zone surface cuts the subject plane and creates a curve of intersection. That curve of intersection must then fit inside the tolerance zone of the theoretical gage.

**Total runout**

Total runout requires features to be cylindrical in form and coaxial with respect to the axis of a feature selected as datum to within a specified tolerance value. Figure 3-36 shows the roller assembly of a machine. For functional reasons it is important to have a precisely uniform distance between this roller and an opposing roller or plate as the roller turns. This type of precision requires total runout specification which controls the combined tolerances of cylindricity and coaxiality.
Figure 3.6. The roller surface in this assembly should be coaxial with the axis of rotation.

Figure 3-37 shows the total runout specification for the outer cylinder.

Figure 3-37. An example of total runout tolerance statement

Datum-A is a high precision bearing fit hole. Datum-B is a bearing hole functionally identical to datum-A but on the other side of the roller. The theoretical gage for total runout is similar to the circular runout except that the gage is three-dimensional. Figure 3-38 shows the theoretical gage for total runout.
Theoretical gage for total runout consists of two concentric cylinders and an axis. During gaging, the datum axis of the gage is aligned with the axis of the datum feature using collapsing cylinders. The cylindrical subject feature is then checked against the tolerance zone. Total runout specification is also easy to verify using the simple inspection set up used for circular runout as shown in Figure 3-39.

Compared to circular runout, the only difference is that once the dial is set in the beginning, the full indicator movement (FIM) includes the range of movement as the indicator is moved to other cross-sections. The total runout is measured as the FIM of the dial. Total runout applied to a shaft shoulder is identical to perpendicularity tolerance with respect to the datum axis.
3.8. Profile tolerances

A surface profile is the general term used to refer to any type of surface or any selected group of surfaces. A profile feature’s size, form, orientation, and location can be controlled using one or more profile statements. The surface profile statement is a good replacement for the traditional ± tolerances. The ± tolerances should still be used for size tolerances with features-of-size in direct fit applications.

Profile of a surface tolerance

Profile of a surface tolerance is the general-purpose surface feature control specification of the GDT standard. Profile tolerance can be applied to surface features of any shape including any grouping of features. In fact, the entire outer skin of a part is an important profile feature that is commonly controlled in default tolerancing.

The profile of a surface tolerance controls one or more surface features to remain within a thin envelope around their basic or theoretical shape. This thin envelope is the tolerance zone geometry. The profile tolerance is often used with a datum frame to control both the size and the location of a profile. Figure 3-40 shows two examples. The circle attached to the arrow is the “All Around” symbol indicating that the tolerance statement applies to all the surfaces making up the inner profile, or the outer profile of the turning support part. The dimensions are theoretical dimensions and are shown here to explain the gage geometry.

Figure 3-40. Examples of profile of surface tolerance specifications
The tolerance statement controls the size of the entire shape. The feature is also located and oriented relative to the bottom surface (datum-A) and it is centered relative to the slot center plane (datum-B). The theoretical gage for this specification is shown in Figure 3-41. The 0.3 mm tolerance zone has a depth larger than the thickness of the part.

Figure 3-41. Theoretical gage for profile statement

The profile tolerance can also be specified to be unequally divided about the theoretical profile geometry using a simple notation. Figure 3-42 shows an example. If the profile tolerance is to be divided such that 0.1 mm is allocated in the direction of adding material to the theoretical shape, the tolerance value of 0.3 U 0.1 will be specified. The first number is the total thickness of the tolerance zone and the second number is the thickness of the portion that adds material to the part. The dashed line in the figure represents the theoretical profile shape.
Figure 3-42. An example of unequally divided profile tolerance specification and the resulting tolerance zone

The specification 0.3 U 0.3 defines the entire tolerance zone in the direction that adds material to the part. The specification 0.3 U 0 defines the entire tolerance zone in the direction that removes material from the part.

Composite profile statements can be used to control various geometric aspects of a profile with finer levels of precision. Composite profile tolerances use the single segment format in which one profile tolerance symbol is shared by multiple statements. Each statement must use a progressively smaller tolerance value. For example, consider the specification in Figure 3-43.
Figure 3-43. An example of a composite profile tolerance with two statements

The upper profile statement controls the size, form, orientation, and location of the feature with an accuracy of 0.3 mm relative to datum-A and datum-B. The lower statement controls the size and orientation relative to datum-A with the finer accuracy of 0.2 mm. You can extend the dining table exam to profile tolerances too. The tabletop replaces the pattern of feet.

The theoretical gage for this statement uses the same datum simulators for both the upper and the lower statements. Once the upper statement is verified, the 0.3 mm tolerance zone is removed and a new 0.2 mm tolerance zone is brought in without changing the part alignment to datum-A simulator. The new tolerance zone is parallel to the datum-A surface. If it is important to control just the size of the profile with finer accuracy a third row of specification can be added with a smaller tolerance value and without any datum reference.

All profile tolerances control the profile size by creating tolerance zones that are equally or unequally disposed about the theoretical profile. The only exception is with the control of conical surfaces in which dimension limits can be used to control the cone size and a profile tolerance can be used to control the cone form. It is important, however, to be able to control profile form separately from profile size for all features without having
to use dimension limits. To do this, the word “FORM” can be placed next to the tolerance statement that is to be interpreted as form control. The following note should be added in the drawing notes section to explain the meaning of the tolerance statement:

FORM: The actual profile must lie within a tolerance zone defined by a pair of profile surface offsets with a separation equal to the tolerance value.

Figure 3-44 shows a specification example. The form control statement can also be added as a third statement in the composite statement format.

![Figure 3-44. Adding a statement for profile form control](image)

For profile form control shown in the figure, the tolerance zone has a thickness of 0.1 mm but the zone size can expand or contract within the limits of size similar to other form control specifications such as cylindricity. If the form control is applied to the lower statement with 0.2 mm tolerance value then the profile size would be dictated by the 0.3 mm zone while the profile combined orientation and form remain within the smaller 0.2 mm zone.

### 3.9. Profile tolerances in place of ± tolerances

When the distance between two parallel planes is to be controlled the use of profile of surface tolerance is preferred to the use of ± limits on dimensions unless the feature is a fit feature (it has opposing planes and fits a mating feature). Consider the shaft in Figure
3-45. The 20 mm distance between two of the shaft shoulders needs to be controlled. In this specification, the indicated feature dimension is to be between 19 and 21 mm.

![Diagram showing the distance between the two shoulders controlled using dimension limits.](image)

**Figure 3-45. The distance between the two shoulders is controlled using dimension limits**

The meaning of the dimension limits, however, is not clear when the two faces do not have perfect forms, or are not perfectly parallel. What is “obvious” to one person is not so obvious to another. Figure 3-46 shows the alternative profile tolerance statement. The figure also shows the theoretical gage resulting from this statement that allows the two planes be as far as 21 mm or as close as 19 mm similar to the dimension limits. The profile tolerance alternative is preferred because its interpretation in terms of an acceptance gage is well defined.

![Diagram showing the control of distance between two planes using profile of surface tolerance statement.](image)

**Figure 3-46. Control of distance between two planes using profile of surface tolerance statement**
When controlling the distance between two planes using a profile statement, it is preferred to specify an origin of measurement. This is shown on the upper part of Figure 3-47. The theoretical gage for this specification is shown in the lower part of the figure.

**Figure 3-47.** Preferred way to control of distance between two planes using profile of surface tolerance statement with specification of origin of measurement

Figure 3-48 shows a shorthand notation for profile specification that does not explicitly label the datum. The end of the dimension line without an arrow implicitly defines a datum plane. This method of controlling distances is as convenient to use as ± limits on dimensions.

**Figure 3-48.** Shorthand specification of profile tolerance with implied datum reference
Profile tolerance can also be applied to control a pattern of features similar to feature patterns controlled by position tolerance statements.

**Profile of a line tolerance**

Profile of a line tolerance is the general-purpose 2D tolerance control specification of the GDT standard. The profile of a line tolerance can be applied to curve features of any shape that result from the intersection of a surface profile and a cutting plane. Profile of a line tolerance can control the size and location of 2D cross-sections of a feature. This statement is often used to control the size of a feature’s cross-sections with finer accuracy after a surface profile tolerance is applied. The theoretical gage for profile of a line statement is a thin envelope around the basic geometric shape of the feature in the plane of cross-sections. Figure 3-49 shows an example of the use of profile of a line tolerance.

![Figure 3-49. An example of the profile of a line statement](image)

In this example, the designer has used the profile of a surface statement to control the size and location of the entire surface feature with a relatively wide 1.2 mm tolerance value. To control the size of each cross-section with more accuracy, the designer adds a profile of a line tolerance with a tolerance value of 0.5 mm. The datum-A reference requires the cutting planes to be parallel to datum-A. If the datum-A is not mentioned, the cutting planes would be normal to the feature spine. The combined effect of the two profile statements is that the bar cross-sections are more precise in size but the bar can have a larger degree of bend or twist as long as it remains within the wider profile of a surface specification.
Exercise Problems

1. Consider the turning support part shown in Figure 3-50.

Figure 3-50. Turning support part

Specify the following labels and tolerances.

a. Define P₁ as datum-A.

b. Define S₁ as datum-B.

c. Define P₂ as datum-C.

d. Make datum-A feature flat to within 0.02 mm.

e. Make datum-B feature perpendicular to datum-A feature to within 0.05 mm.

f. Make datum-C feature perpendicular to datum-A and datum-B to within 0.1 mm.

g. Specify the size limits of H₁ to be 20 ±0.02 mm.

h. Specify the size limits of S₁ to be 76.00 – 76.15 mm.

i. Specify the size limits of H₂ to be 12.00 – 12.25 mm.

j. Specify H₁ to be parallel and located to P₁ and aligned with S₁. Use a tolerance value of 0.1 mm.
k. Control H₂ to be perpendicular to P₁ and aligned to S₁. Use a tolerance value of 0.75 mm and make the specification \( M \) modified.
l. Control the orientation of H₁ to be more precisely parallel to P₁. Use a tolerance value of 0.06 mm.
m. Control the size and location of Prof₁ to be parallel to and located to P₁ and aligned with S₁. Use a tolerance value of 1 mm.
n. Refine the size and orientation precision of Prof₁ relative to P₁ using a tolerance value of 0.5 mm.
o. Refine the form and orientation precision of Prof₁ relative to P₁ using a tolerance value of 0.3 mm.
p. Refine the form precision of Prof₁ using a tolerance value of 0.1 mm.
q. Specify a plate thickness tolerance of 0.5 mm using short-hand profile tolerance specification using any of the two parallel faces as implicit datum.
r. Control the location and orientation of the H₂ holes with respect to P₁ and slot center plane S₁. Use \( M \) modifier for the position control.

2. Consider the part shown in Figure 3-51. For each tolerance statement in the following list, draw the theoretical gage and indicate the gage dimensions.
   a. The position tolerance specification. Create a theoretical gage with two different tolerance zones, one for the upper and one for the lower segments.
   b. Theoretical gages for part (a) when the \( M \) modifier is used in the tolerance statements.
   c. The profile statement for the inner shape.
   d. The profile statement that controls the plate thickness.
3. Consider the part shown in Figure 3-52.
   a) Draw the theoretical gage for the composite position tolerance.
   b) Draw the theoretical gage when the \( \text{M} \) modifier is used with the tolerance value.
   c) Draw the theoretical gage when the \( \text{M} \) modifier is used both with the tolerance value and with datum-B.
   d) Create the CAD model of a physical gage for part c).
4. Consider the part shown in Figure 3-53.
   a) Draw the theoretical gage for the composite position tolerance.
   b) Draw the theoretical gage when the \( \mathcal{M} \) modifier is used with the tolerance value.
   c) Draw the theoretical gage when the \( \mathcal{M} \) modifier is used with the tolerance value, datum-B, and datum-C.
   d) Create the CAD model of a physical gage for part c).
5. Consider the part shown in Figure 3-54.
   a) Draw the theoretical gage for the composite position tolerance.
   b) Draw the theoretical gage when the ☐ modifier is used with the tolerance value.
   c) Draw the theoretical gage when the ☐ modifier is used with the tolerance value, datum-B.
   d) Read the standard and find out whether the ☐ modifier can be used with datum-C plane. If the ☐ modifier can be used on datum-C, draw the resulting gage.
   e) Create the CAD model of the physical gage for part d).
6. Consider the part shown in Figure 3-55. Draw the theoretical gages for all tolerance statements. Also, draw the theoretical gages when the $M$ modifier is used with tolerance values.
The symbol CF stands for “continuous feature”. The continuous feature symbol is used for interrupted surface features and designates the entire surface as the subject feature. In the case of this problem the datum feature A is the entire planar surface not just the half the datum label refers to.

7. Consider the 2D part shown in Figure 3-56 and the position tolerance statement applied to the hole on the top of the figure. The dimensions that define the theoretical location of the hole are not shown.
   a) Create a theoretical gage for this tolerance statement.
   b) Create a theoretical gage when the \( \oplus \) modifier is used with the tolerance value.
Consider the manufactured part is shown in Figure 3-57.

c) Is the location of the hole acceptable?

8. Do the previous problem but this time with the datum priority reversed as shown in Figure 3-58.
Consider the same manufactured part shown again in Figure 3-59.

Consider the position tolerance specification shown in Figure 3-60.
Figure 3-60. Position tolerance with datum-B as the slot center plane

a) Create a theoretical gage for this tolerance statement.
b) Create a theoretical gage when the \( \pm \) modifier is used with both the tolerance value and with datum-B.
c) Is the location of the hole acceptable for each specification?

Figure 3-61 shows the manufactured part.

Figure 3-61. The actual part

10. For the specification shown in Figure 3-62
    a) Create a theoretical gage for the tolerance statement.
b) Is the feature acceptable?
c) Create a theoretical gage when both datum-A and datum-B are specified at MMB. Both datum features have zero geometric tolerances associated with them (not shown).

Figure 3-62. Profile tolerance

Figure 3-63 shows the manufactured part.

Figure 3-63. The actual part

11. Convert the dimension limits in Figure 3-64 to profile of surface tolerances. Convert all the vertical dimensions such that they are measured from the lower surfaces. Do
not label or show any datums in the tolerance statements. For horizontal dimensions, use the specification format that does not call for an origin of measurement.

**Figure 3-64. Convert ± tolerancing format to equivalent profile tolerance format**

12. Read the GDT standard and look for a datum modification symbol that requires a datum feature to move and grip the part. Provide an example and explain the meaning of the specification.
4.1. Overview

Default tolerances are tolerance statements that apply to a specific group or all geometric aspects of a part that have not been explicitly tolerated. For example, if a hole size is tolerated but its location is not tolerated, a default tolerance should apply to control the location of the hole. Otherwise, the manufacturing personnel are not responsible for the location of the hole even when the deviations well exceed manufacturing norms. Default tolerances play an important role in assuring that every part is completely tolerated before being sent to manufacturing.

When design engineers set up their drawings or CAD models for making prototypes, they often specify the tolerances for just a few precision features. In such cases, designers accept the outcome of the manufacturing process for non-precision features assuming that reasonable care is exercised during manufacturing. While adherence to adequate workmanship is expected from manufacturing personnel, having no verifiable requirements on the geometry of features is risky and unacceptable for production documentation used to produce thousands of parts. When tolerances are missing, the designer has no legal grounds to dispute the manufacturing quality even when there are gross mistakes in the part production. The use of default tolerances eliminates such risks. Default tolerance statements are verifiable just like other tolerance statements. Default tolerances, however, are only checked when there is an obvious mistake or lack of expected workmanship in the part production.

In general, default tolerances help design engineers by reducing the time it takes to explicitly tolerance every feature and by preventing inadvertent omissions. Every part should have its own custom-made set of default tolerances. It is an unwise practice to rely on company-wide policies, established default tolerances, historical records of tolerancing similar parts, or any other guideline that absolves the designer from carefully selecting the best default tolerances.

4.2. Default tolerancing methods

The common practice in default tolerancing is to place such tolerances in the title block of drawings as ± limits on drawing dimensions and angles. The text “UNLESS OTHERWISE SPECIFIED” or UOS appears along with the specification. A title block tolerancing format similar to what is shown in Figure 4-1 is common.