7 Metrology of Drilling Operations and Drills

The thing I hate about an argument is that it always interrupts a discussion.

G. K. Chesterton English writer (1874–1936)

Metrology of a drilling operation consists of two closely related parts. The first part is the primary part that relates to the various tolerances on the hole being drilled. Therefore, the first logical part of this chapter is devoted to these tolerances, their standard designations in the part drawing, and proper interpretation in part manufacturing. The latter is vital in the selection of the measuring equipment, design of part fixtures and tool layouts, and selection of the proper tool and machining regime.

The second logical part of this chapter is entirely based on its first part and even overlaps the discussed notions/definitions of this part to some extents. However, it relates to the drill metrology considered for the first time in this book. It presents the author’s unique vision of the subject explaining the meaning of this term, significant drill metrological parameters, their proper definitions, inspection/measurements using various measuring equipment, and their proper assigning by the drill drawing.

7.1 INTRODUCTION

Holes, also known as bores, in general are one of the most common features of mechanical components. In precision applications, surface finish, eccentricity, relative position, shape, and diameter of the holes directly relate to the performance and even service life of the component. Besides diametric error, the general term poor surface quality can be further categorized into several different types of problems, including (1) out-of-roundness, (2) out-of-perpendicular, (3) nailheading (bellmouthing), and (4) off-location. Deviation from the permissible tolerances on these features can impede operation and result in premature failure of the system. Therefore, it is important to inspect and validate how the accuracy parameters of machined holes conform to those assigned by the part drawing/manufacturing drawing for the operation. To do this, the following applies:

1. The quality parameters assigned by the drawing should be understood, and thus, appropriate hole-making operation should be selected.
2. The results of bore machining should be inspected using industrial metrology fundamentals, that is, appropriate gages and measuring methodology.

Metrology of drilling operations is based upon geometrical product specification (GPS). The idea of GPS is to give assurance in obtaining the following essential properties of a product:

- Functionality
- Safety
- Dependability
- Interchangeability.
GPS is implemented through a series of standards. GPS standards have been divided into four groups:

2. Global GPS standards (e.g., ISO 1 on the standard reference temperature).
3. General GPS standards.
4. Complementary GPS standards (e.g., technical rules for drawing indications).

At the first design stages of a component, it is drawn to be an ideal, perfectly manufactured object, that is, having perfect form, texture, and size. Manufacturing processes involved in the production of this designed component cause various deviations of the finished component from its ideal conditions. For example, there can be variations in dimension, form, and surface texture. Because these variations can have a great effect on its service performance, it is, therefore, critical that the definitions of geometry are standardized and understood, so that the variation that is inherent in manufacturing processes can be taken into account. To be able to understand the geometrical variations within component parts, a set of requirements have been produced. These requirements are the part of GPS standards that cover small-scale features (surface texture) and large-scale features (sizes and dimensions, geometrical tolerances, and geometrical properties of surfaces) (Figure 7.1).

GPS is an internationally accepted concept covering all of the different requirements indicated in a technical drawing relating to the part geometry, that is, size, distance, radius, angle, form, orientation, location, runout, surface roughness, surface waviness, surface defects, and edges, as well as all related verification principles, measuring instruments, and their calibration. In other words, this system covers specification of the requirements for the micro- and macrogeometry of a part with associated requirements for verification and calibration of related measuring instruments.

As explained in the introduction of ISO 14660-1:1999, geometrical features exist in three worlds:

1. The world of specification, where several representations of the future workpiece are imagined by the designer.
2. The world of the part, the physical world.
3. The world of inspection, where a representation of a given workpiece is used through sampling of the workpiece by measuring instruments.

For a manufacturing/process engineer/planner, it is important to understand the relationship between these three worlds to assure that the manufacturing process/operation meets the quality requirements assigned by the part designer. In other words, proper understanding of the quality requirements assigned

---

**FIGURE 7.1** General structure of GPS.
by the part designer allows him or her to deploy proper machining operation(s) and gages (including measuring equipment and methodologies as well as a suitable quality assurance system).

ISO 14660 defines standardized terminology for geometrical features in each world as well as standardized terminology for communicating the relationship between each world. It is, therefore, of importance that all persons involved in design, manufacturing, and metrology are equipped with knowledge of the requirements of GPS. It is critical that the communication between the relevant engineering departments involved in GPS is clear, precise, and accurate and that each department understands the product design requirements.

As defined by ASME Y14.5M-1994, geometrical dimensioning and tolerancing known as GD&T is an international language used in drawings to accurately describe a part. The language consists of a well-defined set of symbols, rules, definitions, and conventions that can be used to describe the size, form, orientation, and location tolerances of part features. Moreover, GD&T is an exact language that enables designers to say what they mean in a drawing, thus improving product designs. Production uses the language to interpret the design intent, and inspection looks to the language to determine the inspection methodology. By providing uniformity in drawing specifications and interpretation, GD&T reduces controversy, guesswork, and assumptions throughout the manufacturing and inspection process. A manufacturing engineer/process designer/planner, part quality inspector, and many other specialists involved in part production must possess some basic working knowledge of GD&T to truly understand the designer’s intent in terms of part accuracy.

7.2 STANDARD REFERENCE TEMPERATURE

Temperature variation is one of the most significant sources of gaging error. As manufacturing tolerances get tighter and the margin for gaging error gets smaller, it becomes an issue that must be addressed. In practice, ambient temperatures fluctuate from hour to hour and month to month in shop floor environments, and workpieces vary in temperature as the result of operations and/or seasonal ambient variations. Temperature-induced drift is a subtle effect that is often overlooked or ignored.

The most important GPS standard is ISO 1:2002, Geometrical Product Specifications (GPS)—Standard reference temperature for geometrical product specification and verification. This standard defines the standard reference temperature for all dimensional measurements, and all other GPS standards refer to it. GPS standard ISO 1:2002 states: “The standard temperature for geometrical product specification and verification is fixed at 20°C.” What does it mean for manufacturing? First, during manufacture, the component must be measured close to 20°C; how close to 20°C depends on the materials and the tolerances involved. Second, the final inspection should either be made

- In a temperature-controlled room at 20°C
- By comparison with known artifacts of similar materials at a temperature close to 20°C
- By a measuring gage/machine with thermal compensation
- By the operator with manual corrections.

7.3 SMALL-SCALE FEATURES

The actual cross section of the machined surface viewed at high magnification is far different from the ideal flat, cylindrical, or curvilinear surface indicated in a drawing. Geometrically, the surface is seen to have a large number of minute irregularities (peaks and valleys) superimposed on more widely spaced undulations (waviness). Metallurgical examination of hardened steel components will show the transition from an amorphous layer through re-hardened and tempered zones, which finally merge with the original structure obtained by heat treatment. Analysis by sophisticated methods will reveal the presence of tiny cracks and changing residual stress at different depths below the surface (Astakhov 2010b).

All these features of an engineering surface (made on the finished part) constitute the so-called surface integrity that determines to a great extent the behavior of the part in service (Astakhov 2010b).
Thus, for example, surface texture has a significant effect on the frictional and lubricant-retention properties of the surface. Waviness determines whether mating will be accurate so that leakage can be avoided in sealed joints, metallurgical damage affects the wear resistance of the part, and residual stresses influence the fatigue strength of critical parts besides causing harmful deformations. Therefore, it is evident that surface quality is of critical importance and its proper evaluation is of utmost interest to the shop engineer/process planner.

The irregularity of a machined surface is a result of the machining process including complex influence of many parameters of the machining system such as the choice of cutting tool (geometry and material), feed and speed, part configuration and fixture, machine tool, coolant, and environmental conditions. This irregularity consists of high and low spots machined into a surface by the cutting tool. The standard term for these peaks and valleys is the surface texture. Because surface texture defines to a great extent the performance of the surface (e.g., its wear resistance), it is quantified by the part drawing in certain standard ways.

The term surface finish is well known but the concept is understood more in qualitative terms than in quantitative terms. This is evident from the fact that many industries continue to specify finish as rough, good, smooth, glossy, mirror, etc. None of these terms are sufficiently accurate and besides, they tend to convey different meanings to different people. It is in the common interest to adopt standard quantitative designation of surface finish known as its roughness and standard methods of evaluation with appropriate inspection techniques that will eliminate the variable subjective factor.

7.3.1 Definition of Surface Profile, Cutoff (Sampling) Length, and Centerline

The basic term involved is shown in Figure 7.2. The surface profile is usually measured in a direction perpendicular to the lay of the surface—that is, the predominant direction of the scratch marks. As can be seen in Figure 7.2, surface profile is composed of three distinct types of irregularities. The first type of irregularity is a form of error that is usually of a magnitude that can easily be detected by conventional measuring methods. The second type of irregularity consists of waviness with fairly regular spacing that can be attributed to vibrations of the machine. The third type of irregularity consists of closely spaced peaks and valleys superimposed on the first two types of irregularities. These peaks and valleys can be correlated with the shape, size, and motion of the cutting tool.

The international consensus of opinion has been to relate surface texture (commonly referred to as roughness or surface finish) to the height of the closely spaced irregularities over a short length. This length, which is called the cutoff (sampling) length, is specified so that variations due to waviness or form errors are excluded. Since the peaks and valleys have different heights, it is evident that some kind of average height should be determined. To do this, a surface profile should be analyzed. The profile shown in Figure 7.3 is a highly magnified sectional view of the surface. Within the selected cutoff length, it is possible to draw a CL

![Figure 7.2](image-url) Visualizatio of the basic surface microfeatures.
in such a way that the areas of the peaks above the line are equal to the areas of the valleys below the line. This CL serves as the baseline for determination of many surface roughness characteristics.

### 7.3.2 Common Characteristics of Surface Texture (Roughness) Used in Drilling Operations

There are a great number of ISO standards for characterization and measuring surface texture, for example, ISO 1302 Indication of surface texture in technical product documentation; ISO 3274 Surface texture: Profile method—Nominal characteristics of contact (stylus) instruments; ISO 4287 Surface texture: Profile method—Terms, definitions, and surface texture parameters; ISO 4288 Surface texture: Profile method—Rules and procedures for the assessment of surface texture; and ISO 8785 Surface imperfections—Terms, definitions, and parameters.

There are several different methods of surface texture (roughness) measurement in use today. Some of the commonly used methods in drilling operations are shown in Figure 7.4. The method used on any given part depends largely on where in the world the part is manufactured and the measurement parameters the manufacturer and the customer prefer to use. It is not uncommon for different parties involved in the production to use different methods for surface texture (roughness) measurements.

In North America, the most common parameter for surface texture is average roughness ($Ra$). $Ra$ is calculated by an algorithm that measures the average length between the peaks and valleys and the deviation from the mean line on the entire surface within the sampling length (see Figure 7.4). $Ra$ averages all peaks and valleys of the roughness profile and then neutralizes the few outlying points so that the extreme points have no significant impact on the final results. It’s a simple and effective method for monitoring surface texture and ensuring consistency in measurement of multiple surfaces.

The advantages of using $Ra$ are as follows:

- The most commonly used parameter to monitor a production process
- Default parameter in a drawing if not otherwise specified
- Available even in the least sophisticated measuring instruments
- Statistically a very stable, repeatable parameter
- Good for regular (cutting) and for random (grinding) type surfaces
- A good parameter where a machining operation is under control and where the conditions are always the same, for example, cutting tool, speeds, feeds, MWF (lubricant).

The disadvantages of using $Ra$ are as follows:

- Not a good discriminator for different types of surfaces (no distinction is made between peaks and valleys)
- Not a good measure of sealed surfaces.
<table>
<thead>
<tr>
<th>Parameter</th>
<th>Meaning</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Ra</strong></td>
<td>Arithmetical mean deviation. The average roughness or deviation of all points from a plane fit to the test part surface. Available for profile and areal data. The equation for average roughness is as follows:</td>
</tr>
<tr>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>$$Ra = \frac{1}{L} \int_0^L</td>
</tr>
<tr>
<td><strong>Rz</strong></td>
<td>Ten-point height, or the average absolute value of the five highest peaks and the five lowest valleys over the evaluation length. Also known as the ISO 10-point height parameter. The equation for 10-point height is as follows:</td>
</tr>
<tr>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>$$Rz = \frac{(P1 + P2 + \ldots + P5) - (V1 + V2 + \ldots + V5)}{5}$$</td>
</tr>
<tr>
<td><strong>Rz ISO</strong></td>
<td>Average peak-to-valley profile roughness. The average peak-to-valley roughness based on one peak and one valley per sampling length. The single largest deviation is found in five sampling lengths and then averaged. Identical to Rt ISO.</td>
</tr>
<tr>
<td><strong>Rp</strong></td>
<td>Peak. The maximum distance between the mean line and the highest point within the sample. It is the maximum data point height above the mean line through the entire data set.</td>
</tr>
<tr>
<td><strong>Rv</strong></td>
<td>Valley. The maximum distance between the mean line and the lowest point within the sample. It is the maximum data point height below the mean line through the entire data set.</td>
</tr>
<tr>
<td><strong>Rt PV</strong></td>
<td>Maximum peak-to-valley height. The absolute value between the highest and lowest peaks. It is calculated as $$Rt = Rp + Rv$$</td>
</tr>
</tbody>
</table>

**FIGURE 7.4** Some important standard parameters of surface roughness.
In Europe, the most common parameter for surface texture is mean roughness depth ($R_z$). $R_z$ is calculated by measuring the vertical distance from the highest peak to the lowest valley within five sampling lengths and then averaging these distances. $R_z$ averages only the five highest peaks and the five deepest valleys; therefore, extremes have a much greater influence on the final value. Over the years, the method of calculating $R_z$ has changed but the symbol $R_z$ has not. As a result, there are three different $R_z$ calculations still in use and it is very important to know which calculation is being defined before making the measurement. Sometimes, $R_{max}$—maximum peak-to-valley height—is also specified. The applications of these parameters are as follows:

- $R_z$ is more sensitive than $Ra$ to changes in surface finish as maximum profile heights and not averages are being examined
- $R_{max}$ is useful for surfaces where a single defect is not permissible, for example, a seal with a single scratch
- $R_z$ and $R_{max}$ are used together to monitor the variations of surface finish in a production process. Similar values of $R_z$ and $R_{max}$ indicate a consistent surface finish, while a significant difference indicates a surface defect in an otherwise consistent surface.

$R_z$ to $Ra$ conversion is recommended as follows: Standard BS 1134/1-1972: $R_z = (4-7)Ra$; Siemens Co. $R_z = (4-10)Ra$. Actual ratio depends upon the shape of the profile. An exact conversion of the peak-to-valley height $R_z$ and the CL average height $Ra$ can neither be theoretically justified nor empirically proved. For surfaces that are generated by manufacturing methods of the group metal cutting, a diagram for the conversion from $Ra$ to $R_z$ and vice versa is shown in supplement 1 to standard DIN 4768 part 1, based on comparison measurements. Table 7.1 presents some common values according to this standard, parameters $Rp$ and $Rpm$ are also often used. $Rp$, per ISO 4287, is the max height of any peak to the mean line within one sampling length. $Rpm$, the mean leveling depth—per rules of ISO 4288—is an averaging of $Rp$ over 5 cutoffs, according to ASME B46.1-2002. When $Rp$ is calculated over the evaluation length, it is designated as $Rpm$. Note that many surface gages measure $Rpm$ but report the result as $Rp$.

Applications of these parameters are as follows:

- $Rpm$ is useful in predicting bearing characteristics of a surface
- A low value of $Rpm$ and large value of $R_z$ indicates a plateau surface
- The ratio $Rpm/R_z$ quantifies the asymmetry of profile
- $Rpm$ is recommended for bearing and sliding surfaces and surface substrates prior to coating
- $Rp$ is a good parameter to control coating quality.
The surface texture requirement in the drawing applies to the evaluation length. Certain parameters are defined on the basis of the sampling length and others on the basis of the evaluation length (see ISO 4287, ISO 12085, 13565-2, and ISO 13565-3). When the parameter is defined on the basis of the sampling length, the number of sampling lengths constituting the evaluation length is of decisive importance. Therefore, surface finish is related to the closely spaced irregularities only, and other defects like waviness and form error should be disregarded. The visualization of the lengths involved in surface roughness measurements is shown in Figure 7.5.

In surface roughness measurements, the cutoff wavelength (commonly designated as $\lambda_c$) of a profile filter determines which wavelengths belong to roughness and which ones to waviness. Note that this length is selected at the beginning of surface roughness measurement according to Table 7.2, which represents current industrial practice of ignoring the profile filter step and using a sampling length equal to the cutoff wavelength. The sampling length is the reference for roughness evaluation. This length is equal to the cutoff length. Traversing length is the overall length traveled by the stylus of a surface measuring gage known as profilometer. Evaluation length is a part of the traversing length (Figure 7.5) from where the values of the surface parameters are determined.

The cutoff wavelength $\lambda_c$ has to be chosen in such a way as to include a sufficient number of primary irregularities for the purpose of averaging. Evidently, a large cutoff (sampling) length is necessary for drilled surfaces where individual toolmarks are widely spaced, compared to ground or lapped surfaces produced by the overlapping trajectories of tiny abrasive grains.

The foregoing considerations on surface roughness measurements imply that the actual methodology of surface is as shown in Figure 7.5 as an example of $R_z$ measurement. As can be seen,

$$R_z = \frac{1}{5(R_{z_1} + R_{z_2} + R_{z_3} + R_{z_4} + R_{z_5})} \quad (7.1)$$

![Figure 7.5](image)

**FIGURE 7.5**  Meaning of the traversed, evaluation, and sampling lengths.
The maximum roughness depth $R_{max}$ is the largest of the five roughness depths considered in a sense shown in Figure 7.5.

### 7.3.3 Designation of Surface Texture (Roughness) Parameters

Standard ISO 1302 (2002) is rather complicated document so only its part relevant to drilling operation is covered in this section.

#### 7.3.3.1 Basic Symbols

The basic graphical symbols and their explanations are presented in Table 7.3. When complementary requirements for surface texture characteristics have to be indicated, a line shall be added to the longer arm of any of the graphical symbols illustrated as shown in Figure 7.6. For use in the written text of, for example, reports or contracts, the textual indication for (1) is APA, for (2) MRR, and for (3) NMR. These abbreviators stand for APA, any process allowed; MRR, material removal required; and NMR, no material removed.

When the same surface texture is required on all surfaces around a workpiece outline (integral features), represented in the drawing by a closed outline of the workpiece, a circle shall be added to

### Table 7.2

<table>
<thead>
<tr>
<th>Roughness Parameter</th>
<th>$R_z$ (μm)</th>
<th>$R_a$ (μm)</th>
<th>Cutoff $\lambda_c$ (mm)</th>
<th>Sampling/Evaluation Length $l_e/l_m$</th>
</tr>
</thead>
<tbody>
<tr>
<td>Up to 0.1</td>
<td>Up to 0.02</td>
<td>0.8</td>
<td>0.08/0.4</td>
<td></td>
</tr>
<tr>
<td>Over 0.1</td>
<td>Over 0.02</td>
<td>0.25</td>
<td>0.25/1.25</td>
<td></td>
</tr>
<tr>
<td>Up to 0.5</td>
<td>Up to 0.1</td>
<td>0.8</td>
<td>0.8/4</td>
<td></td>
</tr>
<tr>
<td>Over 0.5</td>
<td>Over 0.1</td>
<td>2.5</td>
<td>2.5/12.5</td>
<td></td>
</tr>
<tr>
<td>Up to 10</td>
<td>Up to 2</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Over 10</td>
<td>Over 2</td>
<td>8</td>
<td>8/40</td>
<td></td>
</tr>
<tr>
<td>Up to 50</td>
<td>Over 10</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Over 50</td>
<td>Over 200</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Up to 200</td>
<td>Up to 80</td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

### Table 7.3

<table>
<thead>
<tr>
<th>Symbol</th>
<th>Meaning</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>Basic graphical symbol: may only be used in isolation when its meaning is</td>
</tr>
<tr>
<td></td>
<td>the surface under consideration or when explained by a note.</td>
</tr>
<tr>
<td></td>
<td>Expanded graphical symbol: machining surface with no indication of other</td>
</tr>
<tr>
<td></td>
<td>details; in isolation, this expanded graphical symbol may only be used</td>
</tr>
<tr>
<td></td>
<td>when its meaning is a surface to be machined.</td>
</tr>
<tr>
<td></td>
<td>Expanded graphical symbol: surface from which removal of material is</td>
</tr>
<tr>
<td></td>
<td>prohibited; this expanded graphical symbol may also be used in a drawing</td>
</tr>
<tr>
<td></td>
<td>relating to a manufacturing process to indicate that a surface is to be</td>
</tr>
<tr>
<td></td>
<td>left in the state resulting from a preceding manufacturing process,</td>
</tr>
<tr>
<td></td>
<td>regardless of whether this state was achieved by removal of material or</td>
</tr>
<tr>
<td></td>
<td>otherwise.</td>
</tr>
</tbody>
</table>
the complete graphical symbol illustrated in Figure 7.6, as shown in Figure 7.7. Surfaces should be indicated independently if any ambiguity may arise from the all-around indication.

### 7.3.3.2 Composition of Complete Graphical Symbol

In order to ensure that a surface texture requirement is unambiguous, it may be necessary, in addition to the indication of both a surface texture parameter and its numerical value, to specify additional requirements (e.g., transmission band or sampling length, manufacturing process, surface lay and its orientation, and possible machining allowances). It may be necessary to set up requirements for several different surface texture parameters in order that the surface requirements ensure unambiguous functional properties of the surface.

The mandatory positions of the various surface texture requirements in the complete graphical symbol are shown in Figure 7.8.

The complementary surface texture requirements in the form of surface texture parameters, numerical values, and transmission band/sampling length should be located at the specific positions in the complete graphical symbol in accordance with the following:

1. **Position a—single surface texture requirement.** Indicates the surface texture parameter designation, the numerical limit value, and the transmission band/sampling length (Table 7.2). To avoid misinterpretation, a double space (double blank) should be inserted between the parameter designation and the limit value.
2. **Positions a and b—two or more surface texture requirements.** Indicate the first surface texture requirement at position a. Indicate the second surface texture requirement at position b. If a third requirement or more is to be indicated, the graphical symbol is to be enlarged.

![Figure 7.6](image)

**FIGURE 7.6** Complete graphical symbol: (a) for APA, for (b) MRR, and for (c) NMR.

![Figure 7.7](image)

**FIGURE 7.7** The outline in the drawing represents the six surfaces shown on the 3D representation of the workpiece (the front and rear surfaces not included).

![Figure 7.8](image)

**FIGURE 7.8** Positions (a–e) for location of complementary requirements.
across and upward in the vertical direction, to make rroom for more lines. The positions $a$ and $b$ are to be moved upward when the symbol is enlarged.

3. **Position $c$—manufacturing method.** Indicates the manufacturing method, treatment, coatings, or other requirements for the manufacturing process to produce the surface, for example, drilled, turned, ground, and plated.

4. **Position $d$—surface lay and orientation.** Indicates the symbol of the required surface lay and the orientation, if any, of the surface lay, for example, “$=$,” “$X$,” and “$M$” (see Figure 7.9).

5. **Position $e$—machining allowance.** Indicates the required machining allowance, if any, as a numerical value given in millimeters.

Figure 7.10 shows the control elements in indication of surface texture requirements in engineering drawings. In this designation, the surface texture requirement shall be indicated as a unilateral or bilateral tolerance. When the parameter designation, the parameter value, and the transmission band are indicated, they are understood as a unilateral upper tolerance limit of the parameter in question (16% rule or max-rule limit). If the parameter designation, the parameter value, and the transmission band indicated are to be interpreted as a unilateral lower tolerance limit of the parameter in question (16% or max limit), then the parameter designation shall be preceded by the letter $L$, for example, $L Ra 0.32$.

<table>
<thead>
<tr>
<th>Symbol</th>
<th>Meaning</th>
<th>Direction of tool marks</th>
</tr>
</thead>
<tbody>
<tr>
<td>$R$</td>
<td>Radial lay orientation relative to the center of the surface displaying the surface texture symbol.</td>
<td><img src="Section_A-A" alt="Radial" /></td>
</tr>
<tr>
<td>$\perp$</td>
<td>Perpendicular orientation relative to the surface in the view displaying the surface texture symbol.</td>
<td><img src="Section_A-A" alt="Perpendicular" /></td>
</tr>
<tr>
<td>$X$</td>
<td>Angular lay orientation in both directions relative to the surface in the view displaying the surface texture symbol.</td>
<td><img src="Section_A-A" alt="Angular" /></td>
</tr>
<tr>
<td>$M$</td>
<td>Multidirectional lay orientation relative to the surface in the view displaying the surface texture symbol.</td>
<td><img src="Section_A-A" alt="Multidirectional" /></td>
</tr>
<tr>
<td>$C$</td>
<td>Circular lay orientation relative to the center of the surface displaying the surface texture symbol.</td>
<td><img src="Section_A-A" alt="Circular" /></td>
</tr>
<tr>
<td>$=$</td>
<td>Parallel lay orientation relative to the surface in the view displaying the surface texture symbol.</td>
<td><img src="Section_A-A" alt="Parallel" /></td>
</tr>
<tr>
<td>$P$</td>
<td>Particulate, non-directional, or protuberant lay orientation relative to the surface displaying the surface texture symbol.</td>
<td><img src="Section_A-A" alt="Particulate" /></td>
</tr>
</tbody>
</table>

**FIGURE 7.9** Lay symbols and their meaning.
A bilateral tolerance is indicated in the complete symbol by placing the requirement for the two tolerance limits above each other, the upper specification limit (16% rule or max rule) preceded by $U$ being indicated over the lower specification limit preceded by $L$. Where the upper and lower limits are expressed by the same parameter with different limit values, the $U$ and $L$ may be omitted provided the omission does not leave any doubt. The upper and lower specification limits, however, are not necessarily expressed by means of the same parameter designation and transmission band. Figure 7.11 shows an example.

There are two different ways of indicating and interpreting the specification limits of surface texture: (1) the 16% rule and (2) the max rule defined by standard ISO 4288:1996. The 16% rule is defined as the default rule for all indications of surface texture requirements. This means that the 16% rule applies to a surface texture requirement when a parameter designation is applied (see Figure 7.12). If the max rule is to be applied to a surface texture requirement, $\text{max}$ shall be added to the parameter designation.

Where no transmission band is indicated in connection with the parameter designation, the default transmission band applies to the surface texture requirement (see Figure 7.12 for no transmission band indicated). Certain surface texture parameters do not have a defined default transmission band,
a default shortwave filter, or a default sampling length (long-wave filter). Consequently, the surface texture indication shall specify transmission band, shortwave filter or long-wave filter, to ensure that the surface texture requirement is unambiguous.

To provide assurance that the surface is controlled unambiguously by the surface texture requirement, the transmission band shall be indicated in front of the parameter designation separated from it by an oblique stroke (/). The transmission band shall be indicated by the inclusion of the cutoff values of the filters (in millimeters), separated by a hyphen ("-"), the shortwave filter indicated first, and the long-wave filter second (Figure 7.13).

7.3.3.3 Practical Designation on Tool Drawings

Standard ISO 1302 (2002) allows simplified designation of the surface texture parameter(s) in a part drawing. Figure 7.14 shows an example. Although such a designation is simple and commonly found in the vast majority of drawings, one should understand the requirements set by the designation shown in this Figure. They are as follows:

1. Surface roughness on all surfaces except one:
   a. One single, unilateral/upper specification limit.
   b. \( R_z = 6.1 \ \mu m \).
   c. 16\% rule, default (ISO 4288).
   d. Default transmission band (ISO 4288 and ISO 3274).
   e. Default evaluation length \( (5 \times \lambda_c) \) (ISO 4288).
   f. Surface lay, no requirement.
   g. Manufacturing process involves material removal.

2. The hole with a different requirement has a surface roughness:
   a. One single, unilateral/upper specification limit.
   b. \( R_a = 0.7 \ \mu m \).
   c. 16\% rule, default.
   d. Default transmission band (ISO 4288 and ISO 3274).
   e. Default evaluation length \( (5 \times \lambda_c) \) (ISO 4288).
   f. Surface lay, no requirement.
   g. Manufacturing process involves material removal.

FIGURE 7.13 Indication of transmission band in connection with surface texture requirement: (a) in text and (b) on drawing.

FIGURE 7.14 Example of simplified surface texture designation.
Figure 7.15 shows two other examples. The same as in the previous example, the following parameters are set: a single, unilateral upper specification limit, 16% rule, default (ISO 4288), default evaluation length ($5 \times \lambda_{c}$) (ISO 3274), default transmission band (ISO 4288 and ISO 3274), surface lay, no requirement, and manufacturing process involves material removal.

Designation of surface texture according to American standard ASME/ANSI Y14.5M-2002 is similar to that set by ISO 1302 (2002) with few exceptions. Figure 7.16 shows positions for complementary requirements that are different from those shown in Figure 7.8. As can be seen, the major difference is that if the surface texture is expressed in $Ra$ (as recommended), then it is shown outside the basic symbols with no symbol $Ra$ shown. Figure 7.17 shows examples.

### 7.3.3.4 Preferred Surface Roughness

Typical surface roughness values vary widely depending on the processes employed (Figure 7.18). Even for given process, roughness values depend on a number of factors (Figures 7.19 and 7.20). For instance, drilling shown in Figure 7.21 indicates a range of anywhere from 12.5 to 0.2 μm. This is because the surface texture and its roughness are system parameters that depend on the design and conditions of the components of the drilling system. Major factors influencing the
ultimate roughness value achievable include the following: the mechanical properties of the material itself, cutting speed and feed, drilling tool design and geometry, MWF characteristics, and tool holder. A minor change in any factor may have a profound effect on the roughness of the surface produced. As previously discussed, the process optimality as a system characteristic can be judged by the difference of the theoretical surface finish and that obtained in a given application.
7.3.3.5 Different Methods for Designating Surface Texture

Although many years have passed since the acceptance of the two basic standards for surface texture designation, namely, ISO 1302 (2002) and ASME/ANSI Y14.5M-2002, many machine shops, manufacturing companies, and even countries still use traditional designation adopted many years ago to avoid confusion in their new and old drawings as these old standards are widely accepted and seem to be well understood within a company (country). Table 7.4 shows some common examples. Therefore, it is of importance for any drilling manufacturing specialist to analyze a part drawing before designing a drilling operation in order to understand the real requirements to the surface texture assigned by this drawing as these real requirements define to a large extent the selection/design of the components of the drilling operation, its efficiency, and gaging needed.
Dimensions are a part of the total specification assigned to parts designed by engineering. However, the engineer in industry is constantly faced with the fact that no two objects in the material world can ever be made exactly the same. The small variations that occur in repetitive production must be considered in the design. To inform manufacturing specialists how much variation from exact size...
is permissible, the designer uses a tolerance or limit dimension technique. A **tolerance** is defined as the total permissible variation of size or the difference between the limits of size. **Limit dimensions** are the maximum and minimum permissible dimensions. Proper tolerancing practice ensures that the finished product functions in its intended manner and operates for its expected life.

All bore dimensions applied to the drawing, except those specifically labeled as basic, gage, reference, maximum, or minimum, will have an exact tolerance, either applied directly to the dimension or indicated by means of general tolerance notes. For any directly tolerated decimal dimension, the tolerance has the same number of decimal places as the decimal portion of the dimension (Gooldy 1998).

Engineering tolerances may broadly be divided into three groups: (1) **size tolerances** assigned to dimensions such as length, diameter, and angle; (2) **geometrical tolerances** used to control a hole shape in the longitudinal and transverse directions; and (3) **positional tolerances** used to control the relative position of mating features. Interested readers may refer to Gooldy (1998) and Smith (2002).

Some basic standard definitions useful for further considerations are as follows:

- **Size**—a number expressing, in a particular unit, the numerical value of a linear dimension (ISO 286-1:1988).
- **Basic size**—the nominal diameter of the shaft (or bolt) and the hole. This is, in general, the same for both components.
- **Size tolerance**—the difference between the maximum and minimum limits of size (Note: Size tolerance is an absolute value without sign) (ISO 286-1:1988).
- **Tolerance zone**—the zone contained between two lines representing the maximum and minimum limits of size, defined by the magnitude of the tolerance and its position relative to the zero line (ISO 286-1:1988).
- **Lower deviation**—the difference between the basic size and the minimum possible component size.
- **Upper deviation**—the difference between the basic size and the maximum possible component size.

In a graphical representation of tolerance limits, a straight horizontal line that represents the basic (nominal) diameter of the bore and to which the deviations are of zero deviation is known as the zero line. The positive deviations from the basic size (diameter) are shown above this line and the negative deviations below it. Figure 7.22 shows a graphical representation of the previous definitions.

![Graphical representation of the basic size deviation and tolerance.](image-url)

**FIGURE 7.22** Graphical representation of the basic size deviation and tolerance.
The ISO system of limits and fits (ISO 286-2-2010 GPS—ISO code system for tolerances on linear sizes—Part 2: Tables of standard tolerance classes and limit deviations for holes and shafts) covers standard tolerances and deviations for sizes up to 3150 mm. The system is based on a series of tolerances graded to suit all classes of work from the finest to the coarsest, along with different types of fits that range from coarse clearance to heavy interference. Here, fit is the general term used to signify the range of tightness that may result from the application of a specific combination of tolerances in the design of mating parts.

The system ISO defines 28 classes of basic deviations for bores. These classes are marked by capital letters (A, B, C, ..., ZC). The tolerance zone for the specified dimensions is prescribed in the drawing by a tolerance mark, which consists of a letter marking the basic deviation and a numerical marking of the tolerance grade (e.g., H7, H8, D5). Table 7.5 presents tolerances commonly used in drilling operations. Figure 7.23 shows the graphical representation of common bore tolerances for a bore of 30 mm dia. Among the shown tolerances, H6–H11 are most common for drilling operations.

<table>
<thead>
<tr>
<th>Nominal Sizes</th>
<th>Micrometers</th>
</tr>
</thead>
<tbody>
<tr>
<td>Over 3 6 10 18 30 40 50 65 80</td>
<td>H6 H7 H8 H9 H10 H11 J6 J7 J8</td>
</tr>
<tr>
<td>Tolerance G6</td>
<td>+12 +14 +17 +20 +25 +29 +34 +34</td>
</tr>
<tr>
<td>Tolerance G7</td>
<td>+16 +20 +24 +28 +34 +40 +47 +54</td>
</tr>
<tr>
<td>Tolerance G8</td>
<td>+22 +27 +33 +40 +48 +56 +66 +79</td>
</tr>
<tr>
<td>Tolerance H6</td>
<td>+8 +9 +11 +13 +16 +19 +22 +22</td>
</tr>
<tr>
<td>Tolerance H7</td>
<td>+12 +15 +18 +21 +25 +30 +35 +35</td>
</tr>
<tr>
<td>Tolerance H8</td>
<td>+18 +22 +27 +33 +39 +46 +54 +68</td>
</tr>
<tr>
<td>Tolerance H9</td>
<td>+30 +36 +43 +52 +62 +74 +87 +91</td>
</tr>
<tr>
<td>Tolerance H10</td>
<td>+48 +58 +70 +84 +100 +120 +140 +140</td>
</tr>
<tr>
<td>Tolerance H11</td>
<td>+75 +90 +110 +130 +160 +190 +220 +220</td>
</tr>
<tr>
<td>J6</td>
<td>+5 +5 +6 +8 +10 +13 +16 +16</td>
</tr>
<tr>
<td>J7</td>
<td>+6 +8 +10 +12 +14 +18 +22 +22</td>
</tr>
<tr>
<td>J8</td>
<td>+10 +12 +15 +20 +24 +28 +34 +34</td>
</tr>
</tbody>
</table>
Drills: Science and Technology of Advanced Operations

7.5 LARGE-SCALE FEATURES: GEOMETRICAL TOLERANCES

7.5.1 CONCEPT AND STANDARD SYMBOLS

The purpose of geometrical tolerancing is to describe the allowable deviations from the ideal part geometry of products. A universal language of symbols is set by various standards for geometrical tolerancing, much like the international system of road signs that advise drivers how to navigate the roads. Geometrical tolerancing symbols allow a design engineer to precisely and logically describe part features in a way they can be manufactured and inspected maintaining accuracy needed for part intended performance in service.

When utilizing geometrical tolerancing and dealing with the component’s real geometrical surface, the deviations from the nominal shape, orientation, and location can be either a single or related to a datum feature of geometrical tolerance type (Figure 7.24). The single classification

![Graphical representation of common bore tolerances for a bore of 30 mm dia.](image)

**FIGURE 7.23** Graphical representation of common bore tolerances for a bore of 30 mm dia.

**FIGURE 7.24** Classification of geometrical tolerances.
relates to form tolerances that are not normally related to a datum, for example, roundness. It is possible to look at the profile tolerance types as form tolerances as they do not always relate to a datum.

There are sets of standards for geometrical tolerancing: GPS set by ISO standards and GD&T set by American standards. The objective of these standards is to define requirements and rules for describing the geometrical requirements for part and assembly geometry. Proper application of these standards ensures that the allowable part and assembly geometry defined in the drawing leads to parts that have the desired form and fit (within limits) and function as intended.

The utilization of geometrical tolerancing in drawings is by

- Geometrical references (datum features)
- Feature control frames
- Geometrical characteristics (symbols)
- Tolerance shapes
- Tolerance zones (values).

The symbols for the drawing indications of geometrical tolerances according to ISO 1101, ISO 5459, ISO 286, and ISO 10579 are shown in Figure 7.25, and Table 7.6 shows the symbols for the geometrical tolerance characteristics. Table 7.7 presents a geometrical tolerance reference chart relevant to drilling operations. With the symbols, all kinds of geometrical tolerances can be expressed.

### 7.5.2 Definitions of Basic Terms

There are some important terms used in geometrical tolerancing, which have to be understood thoroughly in order to comply with the requirements of corresponding standards. A feature is a general term applied to a physical portion of a part, such as a surface, hole, or slot. An easy way to remember this term is to think of a feature as a part surface (Krulikowski 1998). The part shown in Figure 7.26 contains seven features: the top and bottom, the left and right sides, the front and back, and the hole surface.

A feature of size (FOS) is one cylindrical or spherical surface, or a set of two opposed elements or opposed parallel surfaces, associated with a size dimension. A key part of the FOS definition is that the surfaces or elements must be opposed (Krulikowski 1998). An axis, median plane, or centerpoint can be derived from a FOS. FOS dimension is a dimension that is associated with a FOS. Figure 7.27 shows some important examples of features of size and features of size dimensions (Henzold 2006).

Tolerance zone is space limited by one or several geometrically perfect lines or surfaces and characterized by a linear dimension, called a tolerance. A geometrical tolerance applied to a feature defines the tolerance zone within which that feature shall be contained. Unless a more restrictive indication is required, for example, by an explanatory note, the tolerated feature may be of any form or orientation within this tolerance zone.

### 7.5.3 Definitions of Geometrical Tolerance-Related Terms

Figure 7.28 shows the structure of geometrical tolerances.

Form deviation is the deviation of a feature (geometrical element, surface, or line) from its nominal form (Figure 7.29). If not otherwise specified, form deviations are assessed over (or along) the entire feature. Form deviations are originated, for example, by the looseness or error in guidances and bearings of the machine tool, deflections of the machine tool or the workpiece, error in the
<table>
<thead>
<tr>
<th>Symbol</th>
<th>Interpretation</th>
</tr>
</thead>
<tbody>
<tr>
<td>3x</td>
<td>Number(s) of tolerated features when more than 1</td>
</tr>
<tr>
<td>( \text{Ø} 0.02 \text{ CZ A} )</td>
<td>Additional indicators (e.g., ( \text{Ø} 12\text{H8} ))</td>
</tr>
<tr>
<td>Datum letter</td>
<td></td>
</tr>
<tr>
<td>Symbol for common zone</td>
<td></td>
</tr>
<tr>
<td>Tolerance value</td>
<td></td>
</tr>
<tr>
<td>Symbol for cylindrical or circular tolerance zone</td>
<td></td>
</tr>
<tr>
<td>Symbol for tolerance characteristic</td>
<td></td>
</tr>
<tr>
<td>Additional indications (i.e., MD)</td>
<td></td>
</tr>
<tr>
<td>Arrow line to the tolerance feature</td>
<td></td>
</tr>
<tr>
<td>Arrow with letter to the tolerance feature</td>
<td></td>
</tr>
<tr>
<td>Datum triangle and datum box</td>
<td></td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Symbol</th>
<th>Interpretation</th>
</tr>
</thead>
<tbody>
<tr>
<td>Section line, generatrix or surface as tolerance feature or as a datum</td>
<td></td>
</tr>
<tr>
<td>Theoretical exact dimension</td>
<td></td>
</tr>
<tr>
<td>Top side as tolerated feature and as datum feature</td>
<td></td>
</tr>
<tr>
<td>Restricted length or area at tolerated feature and at datum feature</td>
<td></td>
</tr>
<tr>
<td>Restricted length or area at tolerated feature lying everywhere</td>
<td></td>
</tr>
<tr>
<td>Datum target symbol</td>
<td></td>
</tr>
<tr>
<td>Datum target size</td>
<td></td>
</tr>
<tr>
<td>Datum feature and datum target number</td>
<td></td>
</tr>
</tbody>
</table>

**FIGURE 7.25** Common symbols.
fixture of the workpiece, hardness deflection, or wear. The ratio between width and depth of local form deviations is in general more than 1000:1.

*Form tolerance* (Figure 7.24) is the permitted maximum value of the form deviation. According to ISO 1101, there are defined form tolerance zones within which all points of the feature must be contained. Within this zone, the feature may have any form, if not otherwise specified. The tolerance value defines the width of this zone (Figure 7.29). Form tolerances limit the deviations of a feature from its geometrical ideal line or surface form. Special cases of line forms with special symbols are straightness and roundness (circularity) (Figure 7.30). Special cases of surface forms with special symbols are flatness (planarity) and cylindricity. Figure 7.31 shows an example of cylindricity tolerance—the surface shall be contained between two coaxial cylinders with a radial distance 0.05 (Henzold 2006).

*Orientational deviation* is the deviation of a feature from its nominal form and orientation. The orientation is related to one or more (other) datum feature(s). The orientational deviation includes the form deviation (Figure 7.29). If not otherwise specified, orientational deviations are assessed over the entire feature. Orientational deviations are originated similarly as form deviations. They are originated also by erroneous fixture of the workpiece after remounting on the machine tool.

*Orientation tolerance* is the permitted maximum value of the orientation deviation. According to ISO 1101, there are defined orientation tolerance zones within which all points of the feature must be contained. The orientation tolerance zone is in the geometrical ideal orientation with respect to the datum(s). The tolerance value defines the width of this zone (Figure 7.29) (Henzold 2006).

<table>
<thead>
<tr>
<th>Symbol</th>
<th>Interpretation (Modifier)</th>
</tr>
</thead>
<tbody>
<tr>
<td>F</td>
<td>Free state</td>
</tr>
<tr>
<td>L</td>
<td>Least material condition (LMC)</td>
</tr>
<tr>
<td>M</td>
<td>Maximum material condition (MMC)</td>
</tr>
<tr>
<td>S</td>
<td>Regardless of feature size (RFS) (ASME Y14.5)</td>
</tr>
<tr>
<td>U</td>
<td>Unsymmetrical tolerance zone (ASME Y14.5)</td>
</tr>
<tr>
<td>E</td>
<td>Envelope requirement</td>
</tr>
<tr>
<td>R</td>
<td>Reciprocity requirement</td>
</tr>
<tr>
<td>T</td>
<td>Target method (ASME Y14.5)</td>
</tr>
<tr>
<td>LE</td>
<td>Allies to line element</td>
</tr>
<tr>
<td>NC</td>
<td>Not convex</td>
</tr>
<tr>
<td>ACS</td>
<td>Any cross section</td>
</tr>
</tbody>
</table>
### TABLE 7.7
Geometrical Tolerancing Reference Chart

<table>
<thead>
<tr>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>Form</td>
<td>Roundness</td>
<td></td>
<td>Yes</td>
<td>No</td>
<td>No</td>
<td>No</td>
<td>No (note 5)</td>
<td>No (note 5)</td>
<td>ISO 12181</td>
</tr>
<tr>
<td>Form</td>
<td>Cylindricity</td>
<td></td>
<td>Yes</td>
<td>No</td>
<td>No</td>
<td>No</td>
<td>No (note 5)</td>
<td>No (note 5)</td>
<td>ISO 12180</td>
</tr>
<tr>
<td>Form</td>
<td>Straightness</td>
<td></td>
<td>Yes</td>
<td>Yes</td>
<td>Yes (note 1)</td>
<td>No</td>
<td>Yes (note 1)</td>
<td>No (note 5)</td>
<td>ISO 12780</td>
</tr>
<tr>
<td>Location</td>
<td>Symmetry</td>
<td></td>
<td>No</td>
<td>Yes</td>
<td>Yes</td>
<td>Yes</td>
<td>No</td>
<td>No (note 5)</td>
<td>ISO 2768</td>
</tr>
<tr>
<td>Location</td>
<td>Concentricity</td>
<td></td>
<td>No</td>
<td>Yes</td>
<td>Yes</td>
<td>Yes</td>
<td>No</td>
<td>No (note 5)</td>
<td>ISO 2768</td>
</tr>
<tr>
<td>Location</td>
<td>Position tolerance</td>
<td></td>
<td>No</td>
<td>Yes</td>
<td>Yes</td>
<td>Yes</td>
<td>Yes</td>
<td>Yes</td>
<td>ISO 1101</td>
</tr>
<tr>
<td>Orientation</td>
<td>Perpendicularity</td>
<td></td>
<td>Yes</td>
<td>Yes</td>
<td>Yes (note 1)</td>
<td>Yes</td>
<td>Yes (note 1)</td>
<td>No (note 5)</td>
<td>ISO 2768</td>
</tr>
<tr>
<td>Orientation</td>
<td>Parallelism</td>
<td></td>
<td>Yes</td>
<td>Yes</td>
<td>Yes (note 1)</td>
<td>Yes</td>
<td>Yes (note 1)</td>
<td>Yes (note 4)</td>
<td>ISO 2768</td>
</tr>
<tr>
<td>Runout</td>
<td>Circular runout</td>
<td></td>
<td>Yes</td>
<td>Yes</td>
<td>Yes (note 1)</td>
<td>Yes</td>
<td>No</td>
<td>No (note 5)</td>
<td>ISO 1101 ISO 2768</td>
</tr>
<tr>
<td>Runout</td>
<td>Total runout</td>
<td></td>
<td>Yes</td>
<td>Yes</td>
<td>Yes (note 1)</td>
<td>Yes</td>
<td>No</td>
<td>No (note 5)</td>
<td>ISO 1101 ISO 2768</td>
</tr>
</tbody>
</table>

**Notes:**
1. When applied to a FOS.
2. Can also be used as a form control without a datum reference.
3. When a datum FOS is referenced with the MMC modifier.
4. When an MMC modifier is used.
5. Automatic per rule #3.
FIGURE 7.26 Examples of features.

FIGURE 7.27 Examples of features of size and features of size dimensions.

FIGURE 7.28 Structure of geometrical tolerances.
Orientation tolerances limit the deviations of a feature from its geometrical ideal orientation with respect to the datum(s). Special cases of orientation with special symbols are parallelism (0°) and perpendicularity (90°) (Figure 7.32). The orientation tolerance also limits the form deviation of the tolerated feature, but not of the datum feature(s). If necessary, a form tolerance of the datum feature(s) must be specified (Henzold 2006).
Locational deviation is the deviation of a feature (surface, line, point) from its nominal location. The location is related to one or more (other) datum feature(s). The locational deviation includes also the form deviation and the orientational deviation (of the surface, axis, or median face) (Figure 7.29). If not otherwise specified, locational deviations are assessed over the entire feature. Locational deviations are originated similarly as size, form, and orientational deviations (Henzold 2006).

Location tolerance is twice the permitted maximum value of the locational deviation. According to ISO 1101, there are defined location tolerance zones within which all points of the feature must be contained. The location tolerance zone is in the geometrical ideal orientation and location with respect to the datum(s). The tolerance value defines the width of this zone (Figure 7.29). Location tolerances limit the deviations of a feature from its geometrical ideal location (orientation and distance) with respect to the datum(s). Special cases of location with special symbols are coaxiality (when tolerated feature and datum feature are cylindrical) and symmetry (when at least one of the features concerned is prismatic) where the nominal distance between the axis or median plane of the tolerated feature and the axis or median plane of the datum feature is zero (Figure 7.33). The location tolerance also limits the orientation deviation and the form deviation of the tolerated feature (plane surface or axis or median face), but not the form deviation of the datum feature(s). If necessary, a form tolerance for the datum feature(s) must be specified (Henzold 2006).

Runout tolerances are partly orientation tolerances (axial circular runout tolerance, axial total runout tolerance) and partly location tolerances (radial circular runout tolerance, radial total runout tolerance). However, according to ISO 1101, they are considered as separate tolerances with separate symbols because of their special measuring method.

**FIGURE 7.32** Orientation tolerances: drawing indications and tolerance zones.
Circular radial runout (Figure 7.34a). In each plane perpendicular to the common datum axis, the profile (circumference) shall be contained between two circles concentric with the datum axis and with a radial distance of 0.05 mm.

Total radial runout tolerance (Figure 7.34b). The surface shall be contained between two cylinders coaxial with the datum axis and with a radial distance of 0.05 mm. During checking of the circular radial runout deviation, the positions of the dial indicator are independent of each other. However, during checking of the total radial runout deviation, the positions of the dial indicator are along a guiding (straight) line parallel to the datum axis. Therefore, the straightness deviations and the parallelism deviations of the generator lines of the tolerated cylindrical surface are limited by the total radial runout tolerance, but not by the circular radial runout tolerance (Henzold 2006).

7.5.4 Datum Features

In the author’s opinion, the datum specification distinguishes a drawing from a picture. This is particularly true for any tool drawing that must begin with the assignment of the proper datum.

Datums are theoretically perfect points, lines, axes, surfaces, or planes used for referencing features of an object. They are established by the physical datum features that are identified in the drawing. Identification of datum features is done by using a datum feature symbol defined by
standard ISO 5459:2011 Geometrical product specifications (GPS)—Geometrical tolerancing—Datums and datum systems. This symbol consists of a capital letter enclosed in a square frame. A leader line extends from the frame to the selected feature. A triangle is attached to the end of the leader and is applied in the appropriate way to indicate a datum feature (Figure 7.35).

The datum may be established by the following:

- One single datum feature (e.g., Figure 7.34)
- Two or more datum features of the same priority as a common datum (e.g., a common axis, as shown in Figure 7.36), indicated by a hyphen between the datum letters in the tolerance frame
- Two or more datum features with different but not specified priorities (indicated by a sequence of datum letters in the tolerance frame without separation) (Figure 7.37). Because such an indication is not unequivocal and may lead to different measuring results for the same workpiece, this indication has therefore been eliminated from ISO 1101 (now labeled as former practice)
- Datum features of different priorities (e.g., three-plane datum system according to ISO5459) (Figure 7.38).

**FIGURE 7.34** (a) Circular radial runout tolerance and (b) total radial runout tolerance: drawing indications and tolerance zones.

**FIGURE 7.35** Datum symbol.
FIGURE 7.36  Datums of same priority as common datums.

FIGURE 7.37  Datums with priorities not specified.

FIGURE 7.38  Datums with different priorities.
Datums with different priorities (order of precedence) are classified as follows (Henzold 2006):

1. The primary datum—datum feature orientated to the minimum requirement relative to the simulated datum feature.
2. The secondary datum—datum feature orientated without tilting relative to the primary simulated datum feature (only by translation and rotation) according to minimum requirement relative to the secondary simulated datum feature. The secondary simulated datum feature is perpendicular to the primary simulated feature.
3. The tertiary datum—this is defined as a feature or features used to complete the coordinate system in relation to the primary and secondary datums.

Where a component is rough machined, datum target points are used to establish a datum. Datum target frames are used to define the points, lines, or areas where the manufacturing locations or measurement points should be defined from, to create the coordinate system. Figure 7.39 shows an example of a datum target frame. This example shows the location zone for manufacture or measurement to be anywhere within the circular area of a diameter of 8 mm with a theoretical center being at 20 mm by 22 mm from the corner. A1 is the reference name (datum feature and datum target number). The target is an area so it is hatched.

### 7.6 BORE GAGING

Bore gaging, already a significant factor in manufacturing, has become increasingly important in metrology because of the growing concern for total quality assurance (TQA). Compared with OD measurement, bore gaging creates more engineering challenges by the very nature of its special role in QA. This is especially true when measuring difficult-to-assess internal features such as splines, threads, and deep bores.

As machine capability increases, so does the demand for dimensional gaging to cope with tighter limits on tolerances and greater complexity of component parts. Manufacturers are concerned not only with making the component to a specific tolerance band but also with size variation from component to component because it gives them better control of their process. To meet this demand, the gage they use must be able to discriminate size variation better than before. Moreover, many manufacturers now require that any in-process or post-process gage records and sends the measured size of a component to a statistical process control (SPC) or data-collection system. This section aims to familiarize manufacturing professionals with the basic concepts of bore gaging.

### 7.6.1 BORE GAGE CLASSIFICATION AND SPECIFICATION

To measure the bore tolerances, modern manufacturing requires the use of gages. A gage is defined as a device for investigating the dimensional fitness of a part for specific function.
Gaging is defined by ANSI as a process of measuring manufactured materials to assure the specified uniformity of size and contour required by industries. Gaging thereby assures the proper functioning and interchangeability of parts; that is, one part will fit in the same place as any similar part and perform the same function, whether the part is for the original assembly or replacement in service (Nee 2010).

Bore gages may be classified as follows:

1. Master gages.
2. Inspection gages.
3. Manufacturer’s gages.
4. Gages that control dimensions.
5. Gages that control various parameters of bore geometry.
6. Fixed limit working gages.
7. Variable indicating gages.
8. Post process gages.
9. In-process gages.

Master gages are made to their basic dimensions as accurately as possible and are used for reference, such as for checking or setting inspection of manufacturer’s gages. Inspection gages are used by inspectors to check the manufactured products. Manufacturer’s gages are used for inspection of parts during production.

Post process gages are used for inspecting parts after being manufactured. Basically, this kind of gage accomplishes two things: (1) it controls the dimensions of a product within the prescribed limitations and (2) it segregates or rejects products that are outside these limits. Post process gaging with feedback is a technique to improve part accuracy by using the results of part inspection to compensate for repeatable errors in the bore manufacturing operations. The process is normally applied to CNC machines using inspection data to modify the part program and on tracer machines using the same data to modify the part template.

In-process gages are used for inspecting parts during the machining cycle. In today’s manufacturing strategy, in-process gages and data-collection software provide faster feedback on quality. Indeed, the data-collection and distribution aspect of 100% inspection has become as important as the gaging technology itself. Software, specifically designed to capture information from multiple gages, measure dozens of product types and sizes, and make it available to both roving inspectors and supervising quality personnel as needed, is quickly becoming part of quality control strategies as an integral part of the measurement systems analysis (MSA). In conjunction with CNC units, in-process gaging can automatically compensate for workpiece misalignment, tool length variations, and errors due to tool wear.

7.6.2 Components of Gage Accuracy

The accuracy of a gage is determined by three factors:

1. Resolution. For a gage to accurately determine whether or not a component is in tolerance, the general rule is that the gage resolution should be near 10% of the tolerance
2. Reproducibility and Repeatability (R&R)
3. Linear accuracy. This is the value of maximum deviation from the true size a gage will be capable of measuring across its entire working range. As this Figure is intrinsically linked to the range of the gage, it means nothing unless considering both the value of deviation and the range together. Many manufacturers mistakenly use the 10% rule that establishes an appropriate resolution and apply it to linear accuracy. This is incorrect because it does not take into account the range of the gage.
7.6.3 Bore Gage Types

In choosing a bore gage, one should first eliminate those types of bore gages that are least appropriate for the application. For example, for a production environment, noncontact measurement techniques, including optical and laser methods, tend to be bulky, relatively expensive, and inflexible when measuring special internal features. Such gaging requires the part to be taken to the gage. Not all types of noncontact systems are suitable for a production environment, and many entail high maintenance costs.

Gages that control dimensions are used to control bore diameter (Nee 2010). These gages can be either post process or in-process gages. Further, these gages can be either fixed limit gages or variable indicating gages. These gages are classified as (1) plug/pin gages, (2) needle gages, (3) chamfer gages and countersink gages, (4) dial caliper gages, (5) dial bore gages, (6) electronic bore gages, (7) hole micrometers/ingages, and (8) telescopic gages.

The detailed description of bore gages and their selection and tolerances was presented by the author earlier (Astakhov 2014).

7.7 DRILL METROLOGY

For centuries of drilling history, drill metrology was not considered seriously as the tolerances on drilled holes were wide open. Primitive hand gages and eyeballing measurements relying more on common sense and experience than on results of accurate measurements were considered common practice in drill metrology. Nowadays, however, with widening use of HP drills and modern drilling systems, the tolerances on drilled holes became the same as the were for reamed or even for ground holes not long ago. The dimensional parameters of drills to make such holes as well as parameters of drill geometry became of prime concern. Therefore, this section for the first time in the drilling literature discusses and explains drill metrology, its parameters, and their importance, defines the proper terms, presents the measuring methodologies, and provides suggestions on the tolerances and their proper specification for common and HP drills.

The drill metrological features can be thought of as the dimensional and geometrical non-datum features, for example, drill diameter and length, point and chisel angles, and the datum features, for example, drill runout.

7.7.1 Drilling Tool Diameter

Although it may appear that the drill diameter and its tolerance are the simplest part of drilling metrology as its selection is based upon the tool layout (see Chapter 1), it is not always the case, as a number of important system considerations not normally considered in the literature in the field should be involved in the proper selection of drill diameter tolerance. This section aims to explain some critical issues in the selection of the proper drill diameter and assigning its tolerance limits.

The obvious answer to the question what should be the drill diameter is it should be equal to the diameter of the hole to be drilled with this drill. Being perfectly correct under the ideal conditions, this answer, however, does not account for the tolerances on the hole, manufacturing tolerance on the drill, shrinkage of the hole after drilling, and the back taper of the drill, that is, that the drill diameter reduces with each successive re-sharpening (see Section 4.5.9.5).

7.7.1.1 Existing Tolerances

Drilling tool diameter tolerances are considered using practical examples. An unprepared user may think that the drill tolerances should correspond to the hole (bore) tolerances shown in Table 7.5 and depicted in Figure 7.23. As bores of H8–H11 tolerances are mostly drilled, the drill tolerances should primarily conform to these tolerances being somewhere within the corresponding tolerance zones. Reality, however, proves otherwise.
Tables 7.8 through 7.10 show tolerances on drills according to most commonly used sources of such tolerances. As can be seen, being a bit courser, the tolerances according to standard ASME B94.11M-1993 for general-purpose HSS twist drills are in the same line as those by standard DIN 338, while tolerances used by one of the major drill manufacturers depend on the drill material, that is, the tolerances on HSS drills are not nearly the same as those on carbide drills.

To understand the problem with drill tolerances, consider a graphical representation of drill tolerances for the most common H10 bore tolerance. Figure 7.40 shows an example for a bore of

### TABLE 7.8

**Drill Diameter Tolerance according to Standard ASME B94.11M-1993 for General-Purpose HSS Twist Drills**

<table>
<thead>
<tr>
<th>Diameter of Drill</th>
<th>Tolerance</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Inches</strong></td>
<td><strong>Millimeters</strong></td>
</tr>
<tr>
<td>From #97 to #81</td>
<td>From 0.15 to 0.33</td>
</tr>
<tr>
<td>Over Ø1–1/8</td>
<td>Over 0.33–3.18</td>
</tr>
<tr>
<td>Over 1/8–1/4</td>
<td>Over 3.18–6.35</td>
</tr>
<tr>
<td>Over 1/4–5/8</td>
<td>Over 6.35–12.70</td>
</tr>
<tr>
<td>Over 5/8–1</td>
<td>Over 12.70–25.40</td>
</tr>
<tr>
<td>Over 1–2</td>
<td>Over 25.40–50.80</td>
</tr>
<tr>
<td>Over 2–3½</td>
<td>Over 50.80–88.98</td>
</tr>
</tbody>
</table>

### TABLE 7.9

**HSS Drill Diameter Tolerances according to DIN 338 Used by Leading Tool Manufacturers (h8 ISO)**

<table>
<thead>
<tr>
<th>Drill Diameter (mm)</th>
<th>Tolerance (μm)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Over 3–6</td>
<td>−0/−18</td>
</tr>
<tr>
<td>Over 6–10</td>
<td>−0/−22</td>
</tr>
<tr>
<td>Over 10–18</td>
<td>−0/−27</td>
</tr>
<tr>
<td>Over 18–30</td>
<td>−0/−33</td>
</tr>
<tr>
<td>Over 30–50</td>
<td>−0/−39</td>
</tr>
<tr>
<td>Over 50–65</td>
<td>−0/−46</td>
</tr>
</tbody>
</table>

### TABLE 7.10

**Common Tolerances on Carbide Drills by Leading Tool Manufacturers (m7 ISO)**

<table>
<thead>
<tr>
<th>Drill Diameter (mm)</th>
<th>Tolerance (μm)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Over 3–6</td>
<td>+16/+4</td>
</tr>
<tr>
<td>Over 6–10</td>
<td>+21/+6</td>
</tr>
<tr>
<td>Over 10–18</td>
<td>+25/+7</td>
</tr>
<tr>
<td>Over 18–30</td>
<td>+29/+8</td>
</tr>
<tr>
<td>Over 30–50</td>
<td>+34/+9</td>
</tr>
<tr>
<td>Over 50–65</td>
<td>+41/+11</td>
</tr>
</tbody>
</table>
Metrology of Drilling Operations and Drills

20 mm nominal diameter. As can be clearly seen, if a drill is made according to standard ASME B94.11M-1993, then the system runout should be 0.061 mm to achieve the middle of the tolerance zone of the bore; if a drill is made to DIN standard, then the system runout should be 0.0585 mm to achieve the middle of the tolerance zone of the bore; and if a modern carbide drill is used, then the system runout should be 0.0315 mm to achieve the middle of the tolerance zone of the bore. The latter is more realistic for new drilling systems where rigid spindles and precision shrink-fit and hydraulic tool holders are used, while the former two are allocable for older drilling systems. In other words, the standards and recommendations were adopted a long time ago, and thus, they do not consider the recent changes in the accuracy of components of the drilling system, while the tool manufacturing companies are much more adaptive to these rapidly changing conditions.

The drilling tool diameter and its tolerance are the system-dependent parameters. For certain particular cases, one of which is considered further, the issue of the selection of the proper drilling tool diameter and its tolerances becomes critical. Therefore, a methodology for calculating the proper drilling tool diameter and its tolerance zone should be available. Such a methodology is considered in the next section.

7.7.1.2 Methodology to Calculate the Drilling Tool Diameter and Its Tolerance Zone

The selection of the proper drilling tool diameter is important when a tight-tolerance hole is to be machined or when the tool that follows the drill is sensitive to the drilled hole size. As an example, this section discusses a methodology of the selection of the proper diameter and tolerances on a tap drill (used to drill a hole for further tapping with a threading tap). Figure 7.41 shows the model to calculate tap drill diameter, its tolerance, and minimum allowable diameter after regrind for metric threads with H tolerance position. In the model shown in Figure 7.41, the following applies:

\[ D_{1,\text{max}} \] is the maximum minor diameter of internal thread (Table 7.11).

\[ D_{1,\text{min}} \] is the minimum minor diameter of internal thread (Table 7.11).

\[ A_{\text{max}}, A_{\text{min}} \] are the maximum and minimum growth (elastic recovery) of the minor diameter on tapping. These depend on the properties of the work material. Table 7.12 shows \( A_{\text{max}} \) for 8%Si aluminum alloys (e.g., A380), while \( A_{\text{min}} \) is approximately equal to \( A_{\text{max}}/2 \) (GOST 19257-73).

\[ d_{1,\text{max}} \] and \( d_{1,\text{min}} \) are maximum and minimum diameters of the tap hole.

**FIGURE 7.40** Example of drill diameter tolerances for a hole of 20H10 dia.
Drills: Science and Technology of Advanced Operations

646

FIGURE 7.41 The model including metric tolerance system for screw threads with H tolerance position.

TABLE 7.11
Internal Metric Thread—M Profile
Limiting Dimensions for M6, M8 and M10 threads

<table>
<thead>
<tr>
<th>Basic Thread Designation</th>
<th>Tolerance Class</th>
<th>Minor Diameter, ( D_1 ) (mm)</th>
</tr>
</thead>
<tbody>
<tr>
<td>M5 ( \times ) 0.75</td>
<td>6H</td>
<td>4.134 to 4.334</td>
</tr>
<tr>
<td>M6 ( \times ) 1</td>
<td>6H</td>
<td>4.917 to 5.153</td>
</tr>
<tr>
<td>M8 ( \times ) 1.12</td>
<td>6H</td>
<td>6.647 to 6.912</td>
</tr>
<tr>
<td>M10 ( \times ) 1.5</td>
<td>6H</td>
<td>8.376 to 8.676</td>
</tr>
</tbody>
</table>

TABLE 7.12
Maximum and Minimum Growth (Recovery) of the Minor Diameter on Tapping for 8%Si Aluminum Alloys

<table>
<thead>
<tr>
<th>Pitch (mm)</th>
<th>0.8</th>
<th>1</th>
<th>1.25</th>
<th>1.5</th>
</tr>
</thead>
<tbody>
<tr>
<td>( A_{max} ) (mm)</td>
<td>0.064</td>
<td>0.080</td>
<td>0.100</td>
<td>0.120</td>
</tr>
</tbody>
</table>

\[ Td_1 = d_{1,max} - d_{1,min} \] is the tolerance on the tap hole.

\[ sr_{max} \] is the maximum drill setting runout.

\[ sr_{min} \] is the minimum drill setting runout.

\[ d_{dr-min} = d_{1-min} - sr_{min} \] is the minimum diameter of the tap drill (cutoff diameter for regrinds) (to be shown in the drawing).

\[ d_{dr-max} = d_{1-max} - sr_{max} \] is the maximum diameter of the tap drill.

\[ Al-d_{dr} = d_{1-max} - d_{1-min} \] is the total allowance for the tap drill diameter variation.

\[ d_{n-dr-nom} \] is the chosen nominal diameter of new tap drill (to be shown in the drawing).

\[ Td_{n-dr-nom} = d_{dr-max} - d_{n-dr-nom} \] is the tolerance on a new drill (to be shown in the drawing).

\[ Al_{reg} \] is the allowance for regrinding.
The methodology allows determining (1) the diameter of a new drill and its tolerance and (2) the minimum diameter of the tap drill (cutoff diameter for regrinds). It includes the following simple steps:

1. **Determination of** $D_{1\text{min}}$ **and** $D_{1\text{max}}$ **using the tolerance zone assigned by the part drawing.** For a given thread size, these are determined from Table 7.8. For example, for M6 × 1 thread, $D_{1\text{min}} = 4.917$ mm and $D_{1\text{max}} = 5.153$ mm. Graphical representation of the obtained diameters shown in Figure 7.42 (similar to that shown in Figure 7.41) makes this and further steps much more transparent helping to avoid numerical errors.

2. **Determination of the maximum and minimum diameter of the machined hole prior recovery.** As follows from Figure 7.41, $d_{1\text{-max}} = D_{1\text{max}} + A_{\text{max}}/2$ and $d_{1\text{-min}} = D_{1\text{min}} + A_{\text{max}}$. For example, for M6 × 1 thread, $A_{\text{max}} = 0.08$ mm (Table 7.8); therefore, $d_{1\text{-max}} = 5.153 + 0.08/2 = 5.193$ mm and $d_{1\text{-min}} = 4.917 + 0.08 = 4.997$ mm as can be seen in Figure 7.42.

3. **Establishing the drilling system runout** (see Section 7.7.5.5) depending upon the particular holder and setting practice used. For example, for the tap drill meant for M6 × 1 thread, $sr_{\text{max}} = 20 \mu\text{m} = 0.02$ mm for a standard drill setting.

4. **Calculating the minimum tool diameter.** In the considered example, the minimum diameter of the tap drill (cutoff diameter for regrinds) is $d_{\text{dr-min}} = d_{1\text{-min}} - sr_{\text{min}}$. The minimum runout can be zero in the most conservative case. For M6 × 1 thread, the minimum diameter is then calculated as $d_{\text{dr-min}} = 4.997 - 0.000 = 4.997$ mm as indicated in Figure 7.42.

5. **Calculating the maximum tool diameter.** In the considered example, the maximum diameter of the tap drill is calculated as $d_{\text{dr-max}} = d_{1\text{-max}} - sr_{\text{max}}$ as follows from Figure 7.41. In the considered example for M6 × 1 thread, $d_{\text{dr-max}} = 5.193 - 0.02 = 5.173$ mm as indicated in Figure 7.42.

6. **Calculating the tolerance on a new tool.** For the tap drill, it is calculated as $Td_{\text{n-dr-nom}} = d_{\text{dr-max}} - d_{\text{n-dr-nom}}$ (Figure 7.41). In the considered example for M6 × 1 thread, $Td_{\text{n-dr-nom}} = 5.173$ mm − 4.997 = 0.176 mm as indicated in Figure 7.42.

7. **Selecting the nominal tool diameter.** For the tap drill, it should be selected in the range of $d_{\text{dr-min}} - d_{\text{dr-max}}$ (Figure 7.41). In the considered example for M6 × 1 thread, special and standard tools are considered. The nominal drill diameter of 5.15 mm is selected for special tool while that of 5.10 mm—for standard.

8. **Assigning tolerance zone and deviations for the drilling tool.** For the considered example for M6 × 1 thread, the tolerance zone of the diameter of the special drill is assigned to be 0.026 mm as shown in Figure 7.42.
Therefore, for a new special HP drill, its diameter should be indicated in the drawing as $\varnothing 5.15 + 0.026/–0$ and the minimum diameter of the tap drill (cutoff diameter for regrinds) as $\varnothing 4.977$ mm. Therefore, the allowance for drill regrinding is $A_{\text{reg}}$ is 0.153 mm as shown in Figure 7.42. In case of the standard drill, ISO tolerance m7 is indicated in catalogs of leading tool manufacturers (e.g., Sandvik Coromant and MAPAL)—for drill diameter of 5.10 mm, it is $+0.016/+0.004$ as shown in Figure 7.42. It means than the maximum diameter of the selected standard drill is 5.160 mm, while its minimum diameter is 5.104 mm. As such, as clearly seen in Figure 7.42, the allowance for drill regrinding $A_{\text{regS}}$ is 0.107 mm.

### 7.7.1.3 Assessment of the Results

Suggested tap drill diameter and tolerance are $\varnothing 5.15 + 0.026/–0$ mm and the minimum diameter of the tap drill (cutoff diameter for regrinds) is $\varnothing 4.977$ mm. According to the old rule, established by manufacturing practice long time ago, the tap drill diameter is selected as the tap size minus the pitch. This rule somehow made its way into ISO 2306-72 (revision date February 28, 2008), according to which the drill diameter for M6 $\times$ 1 is 5 mm with no tolerances assigned. Accounting for this standard, all reference books on tool selection and some tool manufacturers’ catalogs recommend this diameter. If one uses a standard 5 mm drill with the standard tolerance and if this drill is made of a carbide tool material, then according to Table 7.10, the tolerance limits are $+0.016/+0.04$, that is, at the low end of the acceptable limit according to Figure 7.42. Due to back taper, such a drill will be out of the acceptable tolerance after few regrinds. If, however, the drill is made of HSS, its tolerance according to Table 7.9 is $h8$, that is, 0/–0.018 mm. As can be seen, the drill diameter can be well below the acceptable minimum according to Figure 7.42. Moreover, when such a drill is re-sharpened, its diameter becomes even smaller.

Figure 7.43 shows what happens on tapping when a standard carbide tap drill subjected to five re-sharpenings is used. As can be seen, the tap flute is full of chips. Not only cutting threads (two to three first relieved threads) but also practically all the threads participate in cutting, which creates high cutting torque and leads to tap breakage. Even when the tap can manage to complete the full thread, it breaks on retraction (Figure 7.44) as the hole becomes smaller due to bore springback after tapping. Moreover, the chip left in the tap flute causes its recutting and severely damages the thread as can be clearly seen in Figure 7.44.

The problems with tap breakage and poor quality of the threads were solved in the setting of the powertrain plants of one of the world’s largest automotive company. The tap drills having the diameters calculated using the proposed methodology were used for threading holes M5, M6, M8, M10, and M12. A great reduction in the tap breakage, part scrap, and significant improving in thread quality were achieved.

![Figure 7.43](image_url)  
**FIGURE 7.43** Tap can break due to significant amount of chips accumulated in the flute (a) and due to a high tapping torque as practically all threads are involved in cutting (b).
The foregoing considerations suggest that the tolerance on the drill should be calculated carefully rather than relying on multiple tables and recommendations found in various literature reference sources. Many of these recommendations and even standards were developed a long time ago (although recently revised) so that they do not reflect the changes that have occurred in industry over the past decade. In the considered example, the tap drill of 5 mm served its objective when tap drills were used in old machining system characterized by a great system runout (spindle runout + tool-holder runout). As such, the system runout can reach 0.15 mm that overlaps the lack of tap drill diameters. In modern manufacturing with high-speed spindles and precision shrink-fit and hydraulic holders, the system runout normally does not exceed 5–10 μm (0.005–0.010 mm) that creates the problems discussed previously with the diameter of tap drills. Figure 7.45 shows the quality of thread cut when the calculated diameter of the tap drill was used.

**FIGURE 7.44** Showing poor quality of the machined thread and the tap broken on its retraction.

**FIGURE 7.45** Thread cut when the calculated diameter of the tap drill was used.
### 7.7.1.4 Drill Diameters in the Tool Drawing

Figure 7.46 shows how the diameters with tolerances of the steps of a drill are indicated in the drawing. Note that the drill diameters are absolute, that is, a non-datum feature as measured between the drill corners regardless of a particular location of these corners with respect to other features.

### 7.7.2 Shank Diameter

As discussed in Chapters 1 and 5, drills can have a great number of shank designs/configurations. For HP drills, however, cylindrical shanks are recommended because they assure the highest accuracy of drill location in the holder. In the case when the shank is cylindrical, the shank diameter ($d_{sh}$ in Figure 7.46) is the diameter of the cylindrical shank.

Table 7.13 shows the shank diameters according to standard ASME B94.11M-1993 for general-purpose HSS twist drills. They are a way too coarse and, what is more important, are not suitable for modern hydraulic and shrink-fit drill holders. Table 7.14 shows the tolerances of the shank diameter (straight shank drills) used by leading drill manufacturers for modern HP drills.

As the shank is used as the datum feature and as it is installed with high precision in a tool holder, no scribing/etching (e.g., tool number, manufacturer’s logo, order numbers/letters) that compromises these properties should be placed on the shank for HP drills as when such a drill is installed in the high-precision holder for its measurement or presetting, this scribing/etching affects the accuracy of drill location in the drill holder. According to the author’s experience, tool inspectors/presenters often use a diamond file to smooth-out the scribing/etching on the drill shank (e.g., as those shown in Figure 7.47) to achieve the desirable accuracy of inspection/presetting. Figure 7.48 shows an

---

**FIGURE 7.46** Drill diameter and length in a drawing of a drill.

**TABLE 7.13**

Shank Diameter Tolerances (Straight Shank Drills) according to Standard ASME B94.11M-1993 for General-Purpose HSS Twist Drills

<table>
<thead>
<tr>
<th>Diameter of Drill</th>
<th>Tolerance</th>
</tr>
</thead>
<tbody>
<tr>
<td>Inches</td>
<td>Millimeters</td>
</tr>
<tr>
<td>From #97 to #81</td>
<td>From 0.15 to 0.33</td>
</tr>
<tr>
<td>Over #81–1/8</td>
<td>Over 0.33–3.18</td>
</tr>
<tr>
<td>Over 1/8–1/4</td>
<td>Over 3.18–6.35</td>
</tr>
<tr>
<td>Over 1/4–1/2</td>
<td>Over 6.35–12.70</td>
</tr>
<tr>
<td>Over 1/2–2</td>
<td>Over 12.70–50.08</td>
</tr>
</tbody>
</table>

*Source:* Reprinted from ASME B94.11M-1993 by permission of The American Society of Mechanical Engineers. All rights reserved.
example of the proper placement of the tool information. For straight-flute drills, the tool information can also be placed on a side of the chip flute at its end.

### 7.7.3 Overall Length/Flute Length/Shank Length

Overall length (size $L_{oa}$ in Figure 7.46) is the length from the drill point to the end of its shank. Although it may appear to be the simplest part of the drill parameter selection based upon the tool layout (see Chapter 1), it is not always so as a number of important system considerations should be involved in such a selection that is not normally considered in the literature in the field. This section aims to explain some critical issues in the selection of the proper drill length.

The obvious answer to the question “what should be the drill length?” is “it should be equal to the length of the hole to be drilled with this drill + the length of the transition part between the end of the chip flute and the shank + the length of the drill shank as it follows from the tool layout.” Being perfectly

---

### TABLE 7.14
Shank Diameter Tolerance (Straight Shank Drills) Commonly Used by Leading Drill Manufacturers

<table>
<thead>
<tr>
<th>Diameter of Drill (mm)</th>
<th>Tolerance (mm)</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>Plus (+)</td>
</tr>
<tr>
<td>Over 0–3</td>
<td>0.000</td>
</tr>
<tr>
<td>Over 3–6</td>
<td>0.000</td>
</tr>
<tr>
<td>Over 6–10</td>
<td>0.000</td>
</tr>
<tr>
<td>Over 10–18</td>
<td>0.000</td>
</tr>
<tr>
<td>Over 18–30</td>
<td>0.000</td>
</tr>
</tbody>
</table>

---

**FIGURE 7.47** Scribing/etching placed on the tool shank.

**FIGURE 7.48** An example of the proper placement of the tool information.
correct, this answer, however, does not account for drill re-sharpening as the length of the flute (size $L_{fl}$ in Figure 7.46) becomes shorter while the shank length (size $L_{sh}$ in Figure 7.46) remains the same. Moreover, due to back taper, the drill diameter decreases with each re-sharpening so it may happen that this diameter can be undersized (below the low limit allowed by the hole tolerance) in precision drilling. Therefore, the selection of the initial overall length of a drill should account for these two factors, bearing in mind the number of re-sharpenings as this defines the total tool life. When the minimum allowable drill diameter is the major restrictive factor on drill length, for example, for tap drills, this diameter should be included in the drill drawing, and a note in this drawing should clearly state that the drill cannot be used any further if its diameter is less than the indicated minimum allowable drill diameter. While some drill manufacturers tried to include the minimum allowable length, arguing that the minimum diameter is a function of the back taper so that this length can be calculated and checked even with a simple metallic ruler, it is not correct. This is because the amount of back taper can vary significantly so that for the same minimum length, the corresponding minimum diameter may significantly differ.

The second issue in the consideration of the drill length is the minimum allowable flute length. As discussed in Chapter 4 (Figure 4.20), the length $L_{dr-1}$ is called the setting drill length (gage length), and it is measured from the drill corner to a certain reference point (e.g., from the rear datum face of HSK holder). As the drill becomes shorter with each re-sharpening, its shank portion held in the tool holder is decreased as the drill is pushed out in the tool presetting to keep the same gage length. As such, however, there is no way to compensate for the corresponding decrease in the flute length (size $L_{fl}$ in Figure 7.46). Eventually, after a certain number of re-sharpenings, there is always a danger that the flute length $L_{fl}$ can become shorter than the depth of the hole being drilled.

Figure 7.49 shows the consequences of using a drill that has the flute length $L_{fl}$ shorter than the depth of the hole being drilled. Figure 7.49a shows that the part (although it is made of high-strength

![Fracture of the part](a) ![Broken tool](b) ![Chip clogging in the flute](c) ![Shiny shank due to rubbing](d)

**FIGURE 7.49** Consequences of the insufficient chip flute length: (a) fractured part, (b) broken tool, (c) chip clogging the flute, and (d) shiny part of the shank beyond the drill body relief due to rubbing against the wall of the hole being drilled.
aluminum alloy A390) was actually fractured as the formed chip had no room to escape. Tool breakage (Figure 7.49b) was also a result. Figure 7.49c shows that the elements of the formed chip look as welded because chip clogging creates high compressive force. Figure 7.49d shows a shiny part on the tool shank that indicates that the drill flute was completely buried in the hole so the portion of the shank behind the flute having no body relief rubs against the wall of the hole being drilled.

To prevent this from happening, the flute length should always be longer than the depth of the hole being drilled. A question is how much longer. In the author’s opinion, the flute length (size $L_{fl}$ in Figure 7.46) as it is commonly shown in the tool drawings makes sense only in tool manufacturing, while in use, it is rather a misleading feature. This is because it is not a functional length that can assure reliable chip removal because its depth and profile change toward the end of the flute. Instead, the active flute length $L_{fl-a}$, which is shorter than the flute length $L_{fl}$, should be considered. For the twist drill (Figure 7.50a), this length is from the drill point to the place of the flute where it begins to change its depth, while for the straight-flute drill (Figure 7.50b), this length is measured from the drill point to the beginning of the flute recess (washout as it is commonly referred to in tool drawing and the corresponding radius is given $r_{ws}$). The minimum allowable active flute length $L_{fl-a-min}$ should be equal to the maximum depth of the drilled hole plus a length equal to the drill diameter $d_{dr}$. This flute length should be clearly indicated in the drill drawing so that drill is not to be re-sharpened beyond this minimum length.

The next question is about the length of the body relief $L_{br}$ (Figure 7.50). Figure 7.51 shows what happens if the body relief is shorter than the depth of the hole being drilled. As can be seen, severe

![Figure 7.50](image)

**FIGURE 7.50** Additional parameters to standard drill length to be considered: (a) in a twist drill and (b) in a straight-flute drill.

![Figure 7.51](image)

**FIGURE 7.51** Showing what happens if the body relief is shorter that the depth of the hole being drilled.
rubbing of the cylindrical part of the shank took place that ruined the quality of the drilled hole. Therefore, the minimum body relief length \( L_{br-min} \) should always be greater than the maximum depth of the drilled hole. The rule of thumb is to make this length equal to the active flute length \( L_{fl-a} \). This is because the minimum allowable active flute length \( L_{fl-a-min} \) cannot be readily measured directly on the drill, while the body relief length \( L_{br} \) can. As a result, the body relief length \( L_{br} \) can be measured before each successive drill regrind so that the drill is not reground beyond the minimum allowable active flute length \( L_{fl-a-min} \).

Another important issue that came out recently with applications of HP drills concerns the actual drill length as installed in the holder, \( L_{dr-1} \) (Figure 7.50). As discussed in Chapter 4 (Section 4.4.3, Figure 4.20), this length should be kept to its possible minimum to assure drill resistance to the high axial force and drilling torque arising in drilling. It is considered by the tool layout designers of HP drilling operations that this length can be almost the same as the flute length so that the end of the flute is located near the drill holder face. The author’s experience shows that this is not a good idea. Figure 7.52 shows the problem. As can be seen, deep impressions that damage the tool holder are made by the great amount of the chip particular to HP drilling operations. The matter becomes worse when an abrasive work material, for example, a high-silicon aluminum alloy, is drilled. To prevent this from happening, length \( L_{dr-1} \) should be greater than the flute length \( L_{fl} \) by at least 0.8 of drill diameter.

Standards DIN ISO 2768-1 and ASME B94.11M-1993 (Tables 7.15 and 7.16) establish tolerances on the overall length and on the length of the flute. Note that these tolerances are rather coarse as the modern drill grinding machines are capable to maintain approx. 1/10 of these tolerances. As a result, leading drill manufacturers established their own tighter tolerances for HP drills.

### 7.7.4 Datum

The definition and meaning of the datum is discussed in Section 7.5.4. The datums are positioned in the technical drawing differently depending on the specific requirements and functionality of
the feature or features. It is important to identify these requirements so as not to make fundamental errors when manufacturing or measuring the component. For example, care must be taken whether an axis, surface/feature extension, or target datum is chosen as the datum. Figure 7.53 shows a typical example of datum identification in a drawing of drilling tools. In Figure 7.53a, the datum is the axis of the component, while in Figure 7.53b, the datum is a feature extension (surface). As such, considerably different metrologies involved in the inspection of part deviations with respect to the datum are used. If, for example, the datum is a reamer shank axis (the most common datum for drilling tools), then the tool can be located in a precision collet chuck or a hydraulic tool holder, and the runout of the working end of the tool is measured by rotating the tool in a tool setting machine and measuring the maximum and minimum deviation. If, however, the datum is indicated as shown in Figure 7.53b, the surface of the shank is the datum. As such, the shank must be positioned in a precision V-block and rotated while measuring the maximum and minimum deviation of the tool working end. Besides a very few special cases, the latter is incorrect as the tool working datum is

---

**TABLE 7.15**  
Tolerances on the Overall Drill Length according to Standard ASME B94.11M-1993 for General-Purpose HSS Twist Drills

<table>
<thead>
<tr>
<th>Diameter of Drill</th>
<th>Tolerance</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>Inches</td>
</tr>
<tr>
<td>From #97 to #81</td>
<td>From 0.15 to 0.33</td>
</tr>
<tr>
<td>Over #81–1/8</td>
<td>Over 0.33–3.18</td>
</tr>
<tr>
<td>Over 1/8–1/2</td>
<td>Over 3.18–12.70</td>
</tr>
<tr>
<td>Over 1/4–1</td>
<td>Over 12.70–25.40</td>
</tr>
<tr>
<td>Over 1–2</td>
<td>Over 25.40–50.80</td>
</tr>
<tr>
<td>Over 2–3½</td>
<td>Over 50.80–88.90</td>
</tr>
</tbody>
</table>

*Source:* Reprinted from ASME B94. 11M-1993 by permission of The American Society of Mechanical Engineers. All rights reserved.

---

**TABLE 7.16**  
Tolerances on the Flute Length according to Standard ASME B94.11M-1993 for General-Purpose HSS Twist Drills

<table>
<thead>
<tr>
<th>Diameter of Drill</th>
<th>Tolerance</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>Inches</td>
</tr>
<tr>
<td>From #97 to #81</td>
<td>From 0.15 to 0.33</td>
</tr>
<tr>
<td>Over #81–1/8</td>
<td>Over 0.33–3.18</td>
</tr>
<tr>
<td>Over 1/8–1/2</td>
<td>Over 3.18–12.70</td>
</tr>
<tr>
<td>Over 1/4–1</td>
<td>Over 12.70–25.40</td>
</tr>
<tr>
<td>Over 1–2</td>
<td>Over 25.40–50.80</td>
</tr>
<tr>
<td>Over 2–3½</td>
<td>Over 50.80–88.90</td>
</tr>
</tbody>
</table>

*Source:* Reprinted from ASME B94. 11M-1993 by permission of The American Society of Mechanical Engineers. All rights reserved.
the axis of the shank as most modern tool holders use this datum. Unfortunately, many tools in drawings used in the automotive industry have the datum indicated as shown in Figure 7.53b.

7.7.5 **RUNOUT (STRAIGHTNESS)**

7.7.5.1 Concept

Although the general concept of runout is discussed in Section 7.5.3, a need is felt to clarify its particular definition for drilling tools. In drilling tool measurements, runout is often abbreviated as T.I.R. that should be understood as total indicated reading not total indicated runout as commonly thought of in many machine shops and even in the trade and professional reference literature.

To comprehend the concept, one should first consider the starting point, which is the axis of the datum as shown in Figure 7.54a. In this figure, the datum surface is represented by a circle centered on two perpendicular lines. If the circle is rotated around the intersection of these lines in the measuring machine spindle, it would appear to not move as the axis of the datum is assumed to have
a zero runout. Next, a drill is added to the drawing as shown in Figure 7.54b. As shown, the drill shares its center with that of the datum so its runout is zero. It implies that if dial test indicators are used and set to zero at margin 1 and then the drill is turned so that margin 2 is found to be in the place of margin 1, the reading of the indicator is still zero. In other words, a hypothetical cylinder called the margin cylinder formed in drill rotation and represented by the margin circle at the drill periphery corners 1 and 2 as shown in Figure 7.54b shares the same axis with the datum. When one obtains this result in real (less than ideal) manufacturing world, it usually means the indicator is broken, hitting a stop, or not touching the margin as there is always some runout in real drills.

Let us consider what happens if the margin cylinder is not aligned with that of the datum. This situation is shown in Figure 7.55a. The dashed circle is the previous position of the margin circle, that is, its ideal location, and the solid-line circle is the (new) actual location of this circle. The center of the actual margin circle is marked as $C_{dr}$, while that of the datum is $C_{dt}$. A line from $C_{dt}$ through $C_{dr}$ can be always drawn and this line will always be a radius.

Figure 7.55b shows what happens when the datum surface turns in the measuring machine spindle. As can be seen, the center of the margin circle traces out a small circle. This circle goes to zero diameter when the margin circle’s center is aligned with that of the datum. Note that as the spindle turns, the outer side of the drill will define a circle known as the effective diameter $d_{ef}$. The drill runout $\Delta_{dr}$ then can be calculated as $(d_{ef} - d_{dr})/2 = \Delta_{dr}$.

Figure 7.56 shows the original meaning of the term T.I.R. The test indicator set on margin 1 at drill periphery corner and its reading is marked as shown in Figure 7.56a. Then the drill is rotated by $180^\circ$ (for two-flute drill) and a new reading of the indicator is marked as shown in Figure 7.56b. The absolute difference in indicator readings is called T.I.R. Although nowadays the drill runout is rarely measured as shown in Figure 7.56 as noncontact digital measuring machines are widely available for such a purpose, the term T.I.R. is still in common use to indicate the original meaning of runout.

In all previous considerations of the runout measurement, it was assumed that the runout of the datum is zero. To assure that this is the case, modern drill measuring and presetting machines used to inspect/preset modern high-efficiency drills are equipped with high-precision spindles and tool-holding means.

### 7.7.5.2 Importance

For HP drills, runout is the single most important factor affecting tool life as it has the critical effect on machining accuracy and tool life. To quantify this statement, Big Kaiser Co. tested 3 mm dia. HSS and carbide drills (Shin and Dandekar 2012). Each drill was tested under the same conditions,
with only runout changed. Table 7.17 lists test conditions. The wear land of 0.2 mm was selected as the tool life criterion. Figure 7.57 presents the test results which can be summarized as follows:

- Carbide drills have the highest sensitivity to diminished tool life due to runout. Improving runout from 15 to 2 μm tripled tool life of the solid-carbide drill.
- HSS tools were slightly less sensitive than their solid-carbide counterparts to diminished life. Improving runout from 15 to 2 μm produced a 230% improvement in tool life. Through-coolant HSS tools were even less sensitive to diminished tool life, producing only a 160% improvement in tool life.

### 7.7.5.3 Method of Measurement and Tolerancing According to Standards

The drill runout tolerance is assigned the maximum allowable runout (MAR) by various standards. The MARs according to the known standards are rather coarse as they were developed a long time ago when the drill runout itself was a small fraction of the total drilling system runout.

Standard DIN 1414-2 provides a method of measuring drill runout as shown in Figure 7.58. As can be seen, the actual surface of the shank, not its axis, is selected as the datum that, as discussed previously, undermines the actual drill location in any real holder. For drills greater than 2 mm diameter, MAR is calculated as

\[
\text{Runout} = 0.03 + 0.01 \frac{L_{\text{rel}}}{d_{\text{dr}}} \quad (7.2)
\]

Table 7.18 shows some examples.
Standard ASME B94.11M-1993 recommends the selection of MAR (Jobbers length HSS two-flute drills) as follows:

- For drills less than 0.03125 in. (0.8 mm), MAR can be selected according to the manufacturer’s discretion.
- For drills 0.03125 in. (0.8 mm) and larger, MAR can be calculated per the following formula:

\[
\text{Maximum } \text{TIR} = \left( 0.0001316 \times \frac{\text{overall length}}{\text{drill diameter}} \right) + 0.00368 \text{ in.}
\]  

(7.3)

Table 7.19 shows some examples of MAR calculated using Equation 7.3.

Analysis of the data given by Equations 7.2 and 7.3 and Tables 7.18 and 7.19 shows that MARs of the drill runout are excessive. Moreover, MAR increases when the drill diameter decreases, which is in direct contradiction with common drill manufacturing practice.
Tables 7.20 and 7.21 show MARs according to Russian standards for HSS and carbide drills. Indian standard ICS 25.100.30 (2007) states that MAR shall be within 0.02 mm. In the author’s opinion, many existing standards and recommendations where MAR (tolerance on the radial runout) is assigned are severely outdated and thus cannot be used as guidelines for assigning MAR on HP drills. To the first approximation, MAR for Jobbers length carbide drills should not be greater than 0.005 mm for drills from 3 to 10 mm and 0.008 mm for drills over 10–20 mm dia. The problem, however, is often not with drill runout itself as the drilling system runout affects the efficiency of the drilling operation and tool life. Drill runout is only a part of the total system runout.

### 7.7.5.4 Assigning in Drill Drawings

Figure 7.59 shows the proper assignment of MAR in drill drawings. As previously discussed, it consists of the GD&T runout symbol, runout tolerance, and datum information. In the case shown,
the datum selected is the common datum for drilling tools, that is, the axis of the shank. In case of multistage drills, the runout tolerance should be assigned for each stage.

### 7.7.5.5 System Runout

#### 7.7.5.5.1 The Essence

To achieve truly HP drilling, the system (total) runout should not exceed 15 μm. This runout, measured after the drill has been installed into the tool holder and then with the tool holder into the spindle, is the sum of the runouts in four components of the drilling system: the machine spindle + the spindle/tool holder interface + the tool holder/tool interface + the tool itself (Figure 7.60). If the shop uses an existing machine for HP drilling and has no plans to replace/refurbish the spindle, then the

<table>
<thead>
<tr>
<th>Drill Diameter (mm)</th>
<th>Precision</th>
<th>Radial Runout Tolerance (mm)</th>
</tr>
</thead>
<tbody>
<tr>
<td>From 1 to 2</td>
<td>High</td>
<td>0.02</td>
</tr>
<tr>
<td>Over 2–3</td>
<td></td>
<td>0.03</td>
</tr>
<tr>
<td>Over 3–6</td>
<td></td>
<td>0.06</td>
</tr>
<tr>
<td>Over 6–12</td>
<td></td>
<td></td>
</tr>
<tr>
<td>From 1 to 2</td>
<td>Normal</td>
<td>0.04</td>
</tr>
<tr>
<td>Over 2–3</td>
<td></td>
<td>0.06</td>
</tr>
<tr>
<td>Over 3–6</td>
<td></td>
<td>0.06</td>
</tr>
<tr>
<td>Over 6–12</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

**FIGURE 7.59** Assigning runout tolerance as MAR in the drill drawing.

**FIGURE 7.60** Components of the drilling system that contribute to a total runout.
first two contributors of the error are beyond the control of the process designer. However, on new machines designed for HP drilling, either of the two is likely to be the most significant contributor to the system runout. The spindles used in most of machine tools today typically offer concentricity and tool holder location tolerances sufficient to leave ample room in this 20 μm margin, while those meant for HP drilling are 5–10 μm or even better for automotive applications. A drill designed and made for HP drilling applications is not likely to contribute much to the system runout as modern HP drills are manufactured to tight runout tolerances.

That leaves the tool holder clamping interface as the most significant contributor to the system runout. In fact, a wrong tool holder alone may account for more than 20 μm of error. Many shops use tool holders, such as 3-jaw drill chucks, that allow drill runout to exceed 25 μm (0.001”). Extrapolating from the test data shown in Figure 7.57, a solid-carbide drill with runout of 25 μm (0.001”) would produce fewer than 25 holes. A higher-quality chuck, though more expensive, could improve tool life dramatically.

Savings can be measured in cost per hole as it is common practice in the automotive industry. An average price for the 3 mm dia. carbide drills used in the tests (Shin 2012) is $40. With runout of 2 μm (0.00008”), this drill can produce 148 holes, at $0.27 per hole. With runout of 15 μm (0.0006”), the cost per hole nearly triples to $0.80 per hole. As a result, manufacturers willing to accept 0.0006” runout are passing up an opportunity to cut drilling costs by 66%. Note that this simple cost calculation does not include improvements in surface quality and diametric and shape accuracy (including their repeatability) of machined holes. In the automotive industry, the latter can nearly double the calculated cost saving allowing to abolish semi-finished drilling operations.

Static TIR (runout) is a combination of angular (azimuthal) TIR and radial (offset) TIR (Figure 7.61). Angular TIR is the result of a misalignment (skew) between the rotational axis of the cutting tool and the central axis of the collet/spindle system. For example, if a collet tool holder is used, the causes of this type of runout include improper use of set screws in a two-point collet, poorly aligned central collet bore,
worn spindle taper, and debris between the collet and spindle bore tapers. Radial runout is the result of a lateral (parallel) offset between the rotational axis of the tool and the central axis of the collet/spindle system. The most common causes are an offset collet bore and mounting a tool with a shank smaller than the minimum diameter of the collet gripping range. Naturally, measurements of static TIR always include a combination of both angular and radial runouts.

The only measurement of runout that truly represents the real situation in actual drills is taken while the spindle is turning at operating speed as static dial indicator measurements have very little correlation with spindle performance while in operation. The runout measured this way is known as dynamic runout (often referred to as dynamic TIR) might also be a result of dimensional infidelity but can include other factors such as anisotropic (uneven) material density (rotational imbalance), worn-out spindle bearings, dynamic properties of the tool holder/spindle coupling, and rotational resonances. While a static runout of less than 5 μm TIR is achievable on high-precision drilling machine spindles, this information does not provide proper indication of the more relevant runout measurement at operating speeds. Therefore, the dynamic runout should be measured as part of a TQA program. To do this, a noncontact measuring system should be used. One of such systems is the Lion Precision (Minneapolis, MN) Dynamic Runout System. The capacitance-based, noncontact Lion system can carry out dynamic measurements of spindles at speeds in excess of 80,000 rpm.

7.7.5.5.2 Measurement

There are two basic types of total runout in drilling tools: static and dynamic. Static total runout (often referred to as static TIR) is a result of problems with the physical dimensions, or arrangement, of the components of the spindle/tool holder/tool system. While drill runout (TIR) can easily been measured using one of many high-accuracy tool inspection machines available in the market, static total TIR should be measured after the drill has been installed into the tool holder and then into the spindle. In the simplest case, it can be measured similar to its definitions given in Figure 7.56, as shown in Figure 7.62. As the drill has known (and small) runout, a pin gage having the same diameter as the drill is often installed in the tool holder to simplify the measurements. Such a measurement yields a much more accurate result, as the real total TIR is measured instead of that based on two random points represented by the drill margins.

![Visualization of radial TIR (a) and angular TIR (b).](image-url)
7.7.5.5.3 Adjustment

Even a precision tool holder has a runout accuracy of 3–5 μm, and thus, after the drilling tool has been installed into the tool holder and spindle, the total runout is at best 10–12 μm. While acceptable for many drills of normal accuracy, it is excessive for HP drills and combined drills (e.g., those used to replace three-pass operations into one in the automotive industry). Therefore, various compensation procedures are used to reduce this total runout. As the saying goes, accuracy is the sum total of your compensating mistakes.

When the most popular high-precision shrink-fit tool holders are used, there are no means available for compensation of the total runout—it is what it is as they say. Some compensation becomes available when a hydraulic holder is used and the tool(holder assembly is preset in the spindle of a modern presetting machine (e.g., Zoller). The total runout is measured and the tool is rotated to the position where the direction of the maximum runout of the spindle/holder assembly is opposite to that of the tool. Then the tool is locked in the holder in this position. However, such a compensation method has two pitfalls:

1. It is time-consuming. It is a factor in high-throughput production environment where the load on each presetting machine is up to 40 tools per shift.
2. Modern drilling tools are made with small runouts so that this compensation may not be sufficient.

Therefore, a tool holder capable of correcting the total runout can be beneficial.

Compensating tool holders appeared only recently. Figure 7.63 shows the basic idea of adjustable tool holder as seen by ISCAR. This company has introduced a new tool holder designed to easily adjust for radial and angular misalignment. The new GYRO is an adjustable tool holder that can be used on drilling, tapping, and reaming applications. The design of this tool holder allows for a smooth and easy adjustment to account for any radial or angular tool assembly misalignment. It can adjust angularly by as much as 1°, radially by 0.08 in. (approx. 2 mm).

There are a great number of adjustable tool holders appearing almost monthly from the major drilling tool and tool-holder manufacturers. The most recent have the modular design that allows one to interchange the tapers and clamping portion of the tool minimizing the amount of inventory.

![FIGURE 7.63](image_url) The basic idea of adjustable tool holder. (Courtesy of ISCAR Co., Galilee, Israel.)
These tool holders normally have adjustment screws to minimize TIR by turning them using a key in the manner shown in Figure 7.64. Normally, laser-sensor presetting machines are used instead of dial indicators as shown in Figure 7.64 for clarity. The differences between the available adjustable tool holders are in precision, amount of the adjustments, repeatability, sustainability, balancing, etc. More significant issue, however, is the ability of an adjustable tool holder to maintain the set compensation during the whole duration of tool operation in the machine. Note that it can be measured in months for PCD tools.

7.7.6 Point Angle and Lip Height

7.7.6.1 Point Angle Tolerances

Standard DIN 1414-2 assigns tolerance $\pm 3^\circ$ for the point angle (for drills $H$ having $\Phi_p = 118^\circ$ and drills $W$ having $\Phi_p = 130^\circ$). Standard ASME B94.11M-1993 provides the point angles and their tolerances as shown in Table 7.22.

<table>
<thead>
<tr>
<th>Drill Diameter</th>
<th>Included Angle ($)</th>
<th>Tolerance ($)</th>
</tr>
</thead>
<tbody>
<tr>
<td>From 1/16 to 1/2</td>
<td>118</td>
<td>$\pm 5$</td>
</tr>
<tr>
<td>Over 1/2–1 1/2</td>
<td>118</td>
<td>$\pm 3$</td>
</tr>
<tr>
<td>Over 1 1/2–3 1/2</td>
<td>118</td>
<td>$\pm 2$</td>
</tr>
</tbody>
</table>

**TABLE 7.22**
Point Angles and Their Tolerances according to Standard ASME B94.11M-1993

Source: Reprinted from ASME B94.11M-1993 by permission of The American Society of Mechanical Engineers. All rights reserved.
Based on the author’s experience and capabilities of modern drill point grinding machines, the point angle should be application specific and its tolerance should be within ±1° for HP drills.

### 7.7.6.2 Lip Height Tolerances

The idea of drill symmetry is simple: the cutting edges of a drill (the drill point) should be symmetrical with respect to the axis of rotation. It implies that regardless of a particular point angle, the two half-point angles $\phi_1$ and $\phi_2$ (Figure 7.65) must be equal. Similarly, the length of the cutting edges (lips) $L_1$ and $L_2$ should be equal (Figure 7.65). As every feature has a tolerance, the tolerances on the drill’s corresponding features should be assigned to achieve these conditions with the desired accuracy.

To assign the proper tolerance of the drill features to assure the symmetry of the major cutting edges (lips), one needs to understand what happens when these edges are not symmetrical. Figure 7.66a shows a drill having the major cutting edges (lips) of equal length but unequal half-point angles. One cutting edge does most of the cutting. The consequences are oversized machined holes and shortened tool life due to excessive wear of the corner of the loaded cutting edge. Similar results are obtained when the cutting edges are not of the same length as shown in Figure 7.66b. In this case, however, greater (than

**FIGURE 7.65**  Half-point angles and length of the major cutting edges (lips) must be equal.

**FIGURE 7.66**  Showing what happens when the major cutting edges (lips) are not symmetrical: (a) half-point angles are not equal, (b) the edges are not of the same length, and (c) half-point angles and cutting edges lengths are not equal.
in the previous case) hole oversize and wear of one drill corner are normally found. Figure 7.66c shows consequence of a combination of both unequal lengths and half-point angles. Figure 7.67 shows visual consequence of a severe lip height variation. As can be seen, the chips flowing from the flutes are of different appearance.

Standards DIN 1414-1:2006 and ASME B94.11M-1993 do not assign any tolerance on the half-point angle. Rather, to assure symmetry of the major cutting edges (lips), these standards assign tolerances on the so-called relative lip height (widely known as the lip height variation), understood as the difference in indicator reading on the cutting lips at a given distance from the drill axis measured as shown in Figure 7.68. DIN standard 1414-2: 98-06 specifies the point of the measurement to be at the center of the major cutting edge. Both standards provide the same methodology of the measurement of the relative lip height: Rotate the drill in a V-block against a back (front by standard DIN 1414-2) end stop. Measure the cutting lip height variation on a comparator or with an indicator set at a location approximately 75% of the distance from the center to the periphery of the drill (in the middle of the major cutting edges according to DIN standard 1414-2: 98-06). Table 7.23 and Figure 7.69 show the tolerances on the lip height according to ASME B94.11M-1993 and DIN 1414-1:2006, respectively (The Metal Cutting Tool Institute 1989).

According to Russian standards GOST 20698-75 and GOST 17277-71, the lip height variation is assigned through the axial runout of the cutting edges. The tolerances are shown in Tables 7.24 and 7.25. Although the tolerances set by these standards are tighter compared to those shown in Table 7.23 and Figure 7.69, they are still too coarse for HP drills. Having realized this issue, the leading drill manufacturers significantly tightened lip height variation tolerance as modern drill point grinding machines allow

**FIGURE 7.67** Visual consequences of a severe lip height variation—unequal chip appearance.

**FIGURE 7.68** Relative lip height $\Delta_{lh}$ measurement.
this with no additional cost. Unfortunately, there are a number of drill manufacturers that just follow the data shown in Table 7.23 and Figure 7.69.

To appreciate the tolerance on the lip height set by various standards, one needs to understand its effect on the cutting conditions of a drill. Figure 7.70 defines the involved terms. This figure shows by solid lines the theoretical profile of the major cutting edges in the reference plane. This profile is perfectly symmetrical with respect to the axis of the datum so that the cutting extensions of the

### TABLE 7.23
Tolerances on the Lip Height for General-Purpose Two-Flute HSS Drills

<table>
<thead>
<tr>
<th>Drill Diameter Range</th>
<th>Tolerance (Total Indicator Variation)</th>
</tr>
</thead>
<tbody>
<tr>
<td>1/16–1/8 in. inclusive</td>
<td>0.0020 in.</td>
</tr>
<tr>
<td>1.59–3.18 mm inclusive</td>
<td>0.051 mm</td>
</tr>
<tr>
<td>Over 1/8–1/4 in. inclusive</td>
<td>0.0030 in.</td>
</tr>
<tr>
<td>Over 3.18–6.35 mm inclusive</td>
<td>0.076 mm</td>
</tr>
<tr>
<td>Over 1/4–1/2 in. inclusive</td>
<td>0.0040 in.</td>
</tr>
<tr>
<td>Over 6.35–12.70 mm inclusive</td>
<td>0.102 mm</td>
</tr>
<tr>
<td>Over ½–1 in. inclusive</td>
<td>0.0050 in.</td>
</tr>
<tr>
<td>Over 12.70–25.4 mm inclusive</td>
<td>0.127 mm</td>
</tr>
<tr>
<td>Over 1–3½ in. inclusive</td>
<td>0.0060 in.</td>
</tr>
<tr>
<td>Over 25.4–88.90 mm inclusive</td>
<td>0.152 mm</td>
</tr>
</tbody>
</table>

Source: Reprinted from ASME B94. 11M-1993 by permission of The American Society of Mechanical Engineers. All rights reserved.
projections of the major cutting edges intersect at point O on the axis of the datum and the major cutting edges OA and OB are of the same length. The theoretical uncut chip thickness cross section is also shown. This uncut chip thickness cross section is the same for both cutting edges. The profile of a drill having the lip height variation $\Delta_{lh}$ (as presented by standards ASME B94.11M-1993 and DIN 1414-1:2006) or the axial runout $\Delta_{lh-ax}$ (as defined by standards GOST 20698-75 and GOST 17277-71) is shown by the dashed line. As can be seen, the major cutting edges $O_1A_1$ and $O_1B_1$ are not of the same length so that extensions of their projection in the reference plane intersect at point $O_1$, which does not lie on the axis of the datum.

To facilitate understanding of the lip height variation, a model that correlates the uncut (undeformed) chip thickness and lip height variation was developed (Kobayashi 1967). Its modernized version is shown in Figure 7.71. When lip heights of a drill are the same, the cutting edges $OA_0$ and $OB_0$ of this hypothetical drill are the same and so their working conditions represented by the uncut (undeformed) chip thickness are absolutely the same. When this hypothetical drill is rotated by a half revolution (180°) while the cutting feed is applied, its theoretical point designated by O moves into a new position $O_n$ located in the datum (rotational) axis. The major cutting edges $OA_0$ and $OB_0$ become $OA_n$ and $OB_n$ as shown in Figure 7.71.

Because a two-flute drill is considered, it is clear (see Chapter 1) that the distance $OO_n$ is equal to the feed per tooth $f_z$ and the uncut chip thickness (the chip load):

$$h_{D-O} = f_z \sin \varphi$$

(7.4)
Any real drill is manufactured with the lip height difference $\Delta_{lh}$ that results in uneven cutting edges, that is, the length of the cutting edge $O_1A_1$ is not equal to that of $O_1B_1$. According to standards ASME B94.11M-1993 and DIN 1414-1:2006, the lip height difference $\Delta_{lh}$ is assessed by the rotation of the drill by a half revolution (180°) with no feed (against an end stop). After this rotation, the center $O_1$ is found in the location $O_2$ so that the cutting edges $O_1A_1$ and of $O_1B_1$ become $O_2A_2$ and of $O_2B_2$, respectively. As follows from the model shown in Figure 7.71, the distance $A_1B_2$ is equal to the axial runout, $\Delta_{lh-ax}$ (as defined by standards GOST 20698-75 and GOST 17277-71), and the lip height variation defined by standards ASME B94.11M-1993 and DIN 1414-1:2006 is calculated as

$$\Delta_{lh} = \frac{A_1B_2}{\sin \phi} = \frac{\Delta_{lh-ax}}{\sin \phi} \quad (7.5)$$
In the model shown in Figure 7.71, the distance \( e \) designates the variation of the uncut chip thickness by each cutting edge compared to the hypothetical drill when the real drill is rotated by a half revolution (180°). As follows from this model,

\[
e = 0.5\Delta_{lh-ax} \sin \varphi = 0.5\Delta_{lh}
\]  
(7.6)

If \( h_{D-1} \) designates the uncut chip thickness for the cutting edge \( O_1A_1 \) and \( h_{D-2} \) designates the uncut chip thickness for the cutting edge \( O_1B_1 \), then these can be calculated accounting for Equations 7.4 and 7.6 as

\[
h_{D-1} = h_{D-O} + \Delta f_z = f_z \sin \varphi + 0.5\Delta_{lh-ax} \sin \varphi = f_z \sin \varphi + 0.5\Delta_{lh}
\]  
(7.7)

and

\[
h_{D-2} = h_{D-O} - \Delta f_z = f_z \sin \varphi - 0.5\Delta_{lh-ax} \sin \varphi = f_z \sin \varphi - 0.5\Delta_{lh}
\]  
(7.8)

The variation of the uncut chip thickness in one drill revolution is calculated as the difference between \( h_{D-1} \) and \( h_{D-2} \), that is,

\[
\Delta h_{D1-2} = h_{D-1} - h_{D-2} = \Delta_{lh-ax} \sin \varphi = \Delta_{lh}
\]  
(7.9)

As stated previously, in the author’s opinion, the tolerances on the lip height are too coarse for HP drills. The developed model allows to substantiate this statement showing that these tolerances are unacceptable even for usual drills. As an example, let us consider an 18 mm diameter, HSS drill having standard \( \Phi_p = 118° \) meant to drill a medium carbon steel AISI 1045 (HB170). According to recommendations (Astakhov 2011), the cutting feed is selected to be \( f = 0.30 \) mm/rev so that the feed per tooth is \( f_z = 0.15 \) mm/rev. According to DIN 1414-1:2006 (Figure 7.69), the maximum lip height difference for this drill can be \( \Delta_{lh-max} = 0.10 \) mm. The uncut chip thicknesses for the major cutting edges are calculated using Equations 7.7 and 7.8 as

\[
h_{D-1} = f_z \sin \varphi + 0.5\Delta_{lh-max} = 0.15 \sin 59° + 0.5 \cdot 0.1 = 0.18 \text{ mm }
\]  
(7.10)

\[
h_{D-2} = f_z \sin \varphi - 0.5\Delta_{lh-max} = 0.15 \sin 59° - 0.5 \cdot 0.1 = 0.08 \text{ mm }
\]  
(7.11)

As can be seen, when a drill is made with the maximum allowed lip height difference, the desired 50%/50% chip load (force) balance (discussed in Chapter 4) becomes 70%/30%, which provides evidence to the author’s statement about the coarse tolerances on the lip height assigned by the discussed standards.

Table 7.26 presents the recommended axial runout tolerances for HP drills based on the author’s experience and capabilities of modern drill point grinding machines. The implementation practice of HP drills used in advanced powertrain plants in the automotive industry shows the feasibility and cost-efficiency of these tolerances.

<table>
<thead>
<tr>
<th>Drill Diameter (mm)</th>
<th>Axial Runout Tolerance (mm)</th>
</tr>
</thead>
<tbody>
<tr>
<td>From 1 to 3</td>
<td>0.003</td>
</tr>
<tr>
<td>Over 6–20</td>
<td>0.005</td>
</tr>
</tbody>
</table>
When specially requested by a demanding customer, leading drill manufacturers reluctantly set the lip height variation tolerance in a tool drawing in the manner shown in Figure 7.72. Such a representation is incorrect as no datum for measuring this variation is indicated. In the previously discussed methodology for the lip height variation set by standard DIN 1414-2, the surface of the tool shank is used as the datum so it must be clearly indicated in tool drawings.

In the author’s opinion, the axial runout of the major cutting edges (lips) is the best way to control lip height variation and to assign its tolerance in drill drawings. All modern drill inspection and presetting machines actually measure this runout in drill inspection/presetting with respect to the axis of the drill shank. Figure 7.73 shows the proper representation of the lip height variation as the axial runout in tool drawings.

**7.7.7 Web Thickness, Centrality (Symmetry) of the Web, and Flute Spacing**

**7.7.7.1 Web Thickness**

As discussed in Chapter 4 (Section 4.4.3.3), the web diameter known as the web thickness is selected to be \( d_{wb} = (0.125...0.145)d_{dr} \). However, the full web thickness is rarely used at the drill point in modern drills. Rather, web thinning is applied to reduce the web thickness at the drill point. This is achieved by grinding the gashes on the rake faces as discussed in Chapter 4 (Figures 4.74 through 4.76). Therefore, the web thickness is rather a design parameter that should be left to the discretion of drill designers. As a result, many standards including ASME B94.11M-1993 and GOST 20698-75 do not set any requirements/tolerances on this drill feature.

Surprisingly, standards DIN 1414-1 and DIN 1411-2 define the minimum web thickness as shown in Figure 7.74. DIN 1414-1 represents the margin width as a function of the drill diameter as shown in Figure 7.75. Standard DIN 1414-2 discusses the method of its measurement, suggesting to use a caliper, a stylus instrument, or a micrometer.
7.7.7.2 Centrality (Symmetry) of the Web

The web symmetry notion according to standard DIN 1414-1 is shown in Figure 7.76a. According to this designation, it is the radial shift of the web axis with respect to that of the datum, which is, according to the designation provided by this standard, the axis of the shank. Figure 7.76b shows the method of its measurement according to standard DIN 1414-2. As can be seen, the surface of the shank is used as the datum not its axis as per standard DIN 1414-1 (Figure 7.76a). One may wonder why symmetry tolerance shown in Figure 7.76a is used to set the interlocation of two cylindrical surfaces instead of runout or coaxiality. This is because two points of the web circle as measured by an indicator (Figure 7.76b) do not define a circle or cylinder. Figure 7.77 shows the tolerances on the web symmetry according to standard DIN 1414-1.

Standard ASME B94.11M-1993 provides another term and method of measurement for this feature. It is referred to as centrality of the web. The method of measurement is as follows:

\[ d_{wb-min} \]
Rotate the drill in a close fitting bushing. Record the difference in indicator reading of the web at the point as the drill is indexed 180°. Table 7.27 provides tolerances on centrality of web for general-purpose two-flute HSS drills according to standard ASME B94.11M-1993. Table 7.28 shows the tolerances on the web symmetry according to Russian standard 20698-75 for precision HSS drills.

A simple comparison of the tolerances presented in Figure 7.77 and Tables 7.32 and 7.33 shows that the tolerances on centrality of the web set by standard ASME B94.11M-1993 are much tighter than those set by other listed standards. However, the method of measurement of the tolerance on this feature provided by ASME B94.11M-1993 is highly questionable as the incorrect datum is selected and the suggested close fitting bushing is not a well-defined term particularly in the presence of back taper applied to drills.
In the author’s opinion, the standard tolerances on web symmetry are not sufficiently tight for HP drills. The recommended tolerances on web symmetry for HP drills based on the author’s experience and capabilities of modern drill point grinding machines are shown in Figure 7.77. The implementation practice of HP drills used in advanced powertrain plants in the automotive industry shows the feasibility and cost-efficiency of these tolerances.

7.7.7.3 Flute Spacing

Another closely related feature is the flute spacing. Its meaning is shown in Figure 7.78a. According to standard DIN 1414-1, the flute spacing is measured on the face of the drill, as close as possible to the outer corners using a V-block and a dial gage as shown in Figure 7.78b. The methodology is as follows: Place the drill on the V-block so that one major cutting edge touches the gage pin as shown in Figure 7.78b. Set the gage to zero and rotate the drill through 180° so that the other cutting edge is in contact with the gage pin, and take the reading, which is taken as the flute spacing \( A_{sp} \). The deviation is then calculated as \( \Delta_{fs} = A_{sp}/2 \). Figure 7.79 shows the tolerances on the flute spacing according to standard DIN 1414-1.

The method for measuring the flute spacing provided by standard ASME B94.11M-1993 is absolutely the same. Table 7.29 provides tolerances on the flute spacing for general-purpose two-flute HSS drills according to standard ASME B94.11M-1993.

---

**TABLE 7.27**

Tolerances on Centrality of Web for General-Purpose Two-Flute HSS Drills

<table>
<thead>
<tr>
<th>Drill Diameter Range</th>
<th>Tolerance (Total Indicator Variation)</th>
</tr>
</thead>
<tbody>
<tr>
<td>1/16–1/8 in. inclusive</td>
<td>0.0030 in.</td>
</tr>
<tr>
<td>1.59–3.18 mm inclusive</td>
<td>0.076 mm</td>
</tr>
<tr>
<td>Over 1/8–¼ in. inclusive</td>
<td>0.0040 in.</td>
</tr>
<tr>
<td>Over 3.18–6.35 mm inclusive</td>
<td>0.102 mm</td>
</tr>
<tr>
<td>Over ¼–½ in. inclusive</td>
<td>0.0050 in.</td>
</tr>
<tr>
<td>Over 6.35–12.70 mm inclusive</td>
<td>0.127 mm</td>
</tr>
<tr>
<td>Over ½–1 in. inclusive</td>
<td>0.0070 in.</td>
</tr>
<tr>
<td>Over 12.70–25.4 mm inclusive</td>
<td>0.178 mm</td>
</tr>
<tr>
<td>Over 1–3½ in. inclusive</td>
<td>0.0100 in.</td>
</tr>
<tr>
<td>Over 25.4–88.90 mm inclusive</td>
<td>0.254 mm</td>
</tr>
</tbody>
</table>

*Source:* Reprinted from ASME B94.11M-1993 by permission of The American Society of Mechanical Engineers. All rights reserved.

**TABLE 7.28**

Tolerances on the Web Symmetry according to Russian Standard GOST 20698-75 for Precision HSS Drills

<table>
<thead>
<tr>
<th>Drill Diameter Range (mm)</th>
<th>Maximum Asymmetry of the Web (mm)</th>
</tr>
</thead>
<tbody>
<tr>
<td>From 3 to 6 inclusive</td>
<td>0.10</td>
</tr>
<tr>
<td>Over 6–10 inclusive</td>
<td>0.15</td>
</tr>
<tr>
<td>Over 10–20 inclusive</td>
<td>0.20</td>
</tr>
</tbody>
</table>
Table 7.30 presents the tolerances on the flute spacing according to Russian standard 20698-75 for precision HSS drills.

In the author’s opinion, the standard tolerances on the flute spacing (Figure 7.79, Tables 7.29 and 7.30) are not sufficiently tight for HP drills. The problem with this error is that there is no adequate metrology support for the detection of this error. The standard measuring methodologies include the location of a drill in a V-block, so the drill real axis of rotation is not properly defined. Common tool presetting machines allow focusing only on one lip and then rotating the drill to focus on the other.
As such, the flute spacing error known also as the web eccentricity cannot be detected. Existing tool geometry measurement machines do not include this feature in their basic programs. When machining aluminum and its alloys, the web eccentricity, however, can be observed after a short time of cutting. Figure 7.80 shows an example. As can be seen, the two parts of the chisel edge are not equal that causes hole size and location problems. When machining steels, web eccentricity causes the so-called chisel edge walking hindering precise hole location at the tool entrance. Figure 7.81 shows three hole entrances drilled by tools having (1) excessive web eccentricity, (2) allowable (by the standards) web eccentricity, and (3) less than 1° web eccentricity. In practice of CNC machining, to prevent chisel edge walking, a spot (Figure 4.7) or center (Figure 4.8) drill is first used to make hole starts, thus to assure their accurate locations and improve entrance conditions for the drill. In practice of the automotive industry, a short end mill is widely used to start the hole (cut pilot holes) when an inclined hole entrance or uneven/out of location core hole is to be drilled.

The recommended tolerances on web symmetry for HP drills based on the author’s experience and capabilities of modern drill point grinding machines are shown in Figure 7.79.

### 7.7.8 Chisel Edge Centrality

As discussed in Chapter 4, the direction of the chisel edge is fully defined by its angle \( \psi_{ci} \), which is an outcome of many drill geometrical parameters, for example, the clearance angle of the major
cutting edges (lips), type of flank surfaces, and point angle. Ideally, this edge should pass through the center of the drill designated by point 0 in Figure 7.82, which is the point on the axis of the datum feature. In reality, however, it happens rather rarely so actual chisel edges on real drills do not pass through the center 0. In the author’s opinion, a tolerance on the chisel edge centrality $\Delta_{cc}$ in the sense shown in Figure 7.82 should be introduced for HP drills as this feature has almost the same effect as the web eccentricity.

Unfortunately, the existing standards, including DIN 1414-1 and ASME B94.11M-1993, do not define such a feature and thus its tolerance. Table 7.31 presents the recommended tolerances on chisel edge centrality for HP drills.
The definition and importance of application-specific back taper for HP drills are discussed in Chapter 4 (Section 4.5.9.5). It was pointed out that the amount of the back taper for HP drills should be much greater than that for common drills. Moreover, the higher the yield strength of the work material and the lower its modulus of elasticity, the greater back taper should be used.

Standard DIN 1414-1 recommends the back taper to be in the range of 0.02–0.08 mm/100 mm of drill length. Standard DIN 1414-2 suggests measuring it from the outer corners toward the shank of the drill using a micrometer or an indicator. No tolerance is assigned on the back taper.

Standard ASME B94.11M-1993 provides the tolerances per unit of flute length shown in Table 7.32. This unit length is not specified by the standard and the tolerances are too coarse. Probably, the standard assigns the back taper of the drill per 1 in. (25.4 mm) of its length as in this case, the provided data are in good agreement with those set by standard DIN 1414-1.

Table 7.33 shows back taper for drills for machining difficult-to-machine alloys provided by Russian standard GOST 20698-75 for precision HSS drills. As can be seen, the values of the back taper are much greater than those according to the DIN and ASME standards.

The drawbacks of the values of the back taper and its tolerance provided by various standards are as follows:

1. The length of the flute over which the back taper should be applied is not specified.
2. Besides Russian standard GOST 20698-75, the back taper is assigned in a rather wide range with no reference to the properties of the work material, so one may wonder which particular value of the back taper should be used in a particular case of drilling.

<table>
<thead>
<tr>
<th>Drill Diameter Range (mm)</th>
<th>Chisel Edge Centrality (mm)</th>
</tr>
</thead>
<tbody>
<tr>
<td>From 3 to 6 inclusive</td>
<td>0.010</td>
</tr>
<tr>
<td>Over 6–10 inclusive</td>
<td>0.014</td>
</tr>
<tr>
<td>Over 10–20 inclusive</td>
<td>0.018</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Diameter of Drill</th>
<th>Tolerance per Unit of Flute Length</th>
</tr>
</thead>
<tbody>
<tr>
<td>Inches</td>
<td>Millimeters</td>
</tr>
<tr>
<td>From #97 to #81</td>
<td>From 0.15 to 0.33</td>
</tr>
<tr>
<td>Over #81–1/8</td>
<td>Over 0.33–3.18</td>
</tr>
<tr>
<td>Over 1/8–⅓</td>
<td>Over 0.33–6.35</td>
</tr>
<tr>
<td>Over ¼–½</td>
<td>Over 3.18–12.7</td>
</tr>
<tr>
<td>Over ½–1½</td>
<td>Over 12.7–25.4</td>
</tr>
<tr>
<td>Over 1–3½</td>
<td>Over 25.4–88.90</td>
</tr>
</tbody>
</table>

Source: Reprinted from ASME B94.11M-1993 by permission of The American Society of Mechanical Engineers. All rights reserved.
3. Only ASME B94.11M-1993 assigns a tolerance on the back taper. However, the assigned tolerances are too coarse.

4. The measuring of the back taper on the HP PCD (CVD) drills and reamers cannot be accomplished as recommended by standard DIN 1414-2 or even on the modern drill presetting machines because the length of the cutting part of the tool is too short to distinguish the difference in diameters in the presence of even smallest runout. On the other hand, these tools are extremely sensitive to values of back taper. Modern tool geometry inspection machines can handle such measurements provided they are properly programmed and used for this geometrical feature.

According to the author’s experience, the back taper used for HP drills should be $0.08–0.15\text{ mm/100 mm of drill length}$ for machining of carbon and alloyed steels and cast irons, while for work materials of low elasticity modulus and/or high yield strength (e.g., high-silicon aluminum alloys, titanium alloys, heat-resistance alloys), it can be increased up to $0.4\text{ mm/100 mm of drill length}$. In the author’s opinion, the tolerance on the back taper should be approximately $1/8$ of the back taper value selected for a given case of drilling and clearly indicated in the tool drawing.

### 7.7.10 Margin Width

As discussed in Chapter 4 (Section 4.4.3.3), the margin width, $b_m$, for common drills is selected using the following formula:

$$b_m = (0.2...0.5)\sqrt[3]{d_{dr}}$$

(7.12)

As it is a purely design parameter of the drill, many national and international standards do not set any requirement on this design feature. Standard 1414-1:2006-11, however, set such requirements. Figure 7.83 shows the margin width as represented by this standard. Such a representation is

![Margin Width Representation](image-url)

**FIGURE 7.83** Margin width representation according to standard 1414-1:2006-11.
misleading as the actual width of the margin can only be viewed in a cross section perpendicular to
the drill axis as shown in Chapter 4, Figure 4.37.

Figure 7.84 shows margin width and its tolerance (represented by the upper and lower limits) as a function of the drill diameter according to standard 1414-1:2006-11. A simple comparison of these data with those given by Equation 7.12 shows that they are almost the same. As discussed in Chapter 4, such values of the margin width are excessive for HP drills even if the lower limit given by Figure 7.84 is considered. This issue is discussed in Chapter 4 (Section 4.5.9.6).

**7.7.11 Angle of Helix**

The definition of the helix angle for twist drills is discussed in Chapter 4 (Section 4.5.3, Figure 4.48). Standard 1414-1:2006-11 is probably the only standard that establishes the requirements on this angle. Standard 1414-2 provides a measuring procedure for this angle as follows: the helix angle, $\omega_d$ (see Figure 7.85), shall be measured at both minor cutting edges (helical margins, auth.) using a TMM, whose reticle is to be aligned with the minor cutting edge, as shown in Figure 7.85, and the reading of the angle is to be taken. Figure 7.86 shows the lead of helix versus the drill diameter for low helix angle ($H$), standard helix angle ($N$), and high helix angle ($W$) drills and their tolerances.

In the author’s opinion, these data are redundant and obsolete. It is redundant as the lead of helix is calculated (see Equation 4.20) as

$$p_{hl} = \frac{\pi d_{dr}}{\tan \omega_d}$$

(7.13)

It is obsolete because modern drills are ground using CNC machines having virtually zero helix angle error.

Moreover, the measuring procedure for the helix angle is provided by standard 1414-2 as it is not clear how to align the TMM reticle (straight line) and the projection of the helix on the reference
Drills: Science and Technology of Advanced Operations

plane as shown in Figure 7.85. The proper way to measure the helix angle at the drill margin is to set the TMM reticle as the tangent to the projection of the helical surface into the reference plane at the point where this projection intersects the datum axis as shown in Figure 7.85. The problem is that this axis is not visible in measuring with such a microscope so the helical angle can be measured only approximately.

**FIGURE 7.85** Measurement of helix angle.

**FIGURE 7.86** The lead of helix versus the drill diameter.
7.7.12 Clearance Angle

Russian standard GOST 2034-80 recommends the tolerance on the clearance angle to be ±4° for \( d_{dr} < 3 \text{ mm} \) dia., while for \( d_{dr} \geq 3 \text{ mm} \) diameter, this tolerance is ±3° for HSS twist drills. There is no indication in which plane in T-hand-S and how it is measured.

Standard DIN 1414-1:2006-11 does not set any tolerance on the clearance angle. Rather, it provides just its definition in the manner shown in Figure 7.87. Such a definition is incorrect in many respects:

1. The shown angle is not flat as the boundary lines are located in different planes. While in the upper line (boundary), the clearance angle is drawn from the drill corner, the other (lower) line (boundary) is drawn from the top of the chisel edge that undermines the definition of the clearance angle according to any tool geometry standard.

2. The cutting edge is not parallel to the drill transverse axis. Rather, the cutting edge is inclined so that the drill corner lies on this axis. The side angle at the drill corner measured this way can be regarded as the T-mach-S side clearance angle (Chapter 4). However, as discussed in Chapter 4 (Figure 4.66), this is not the clearance angle shown in tool drawings where the T-hand-S side clearance angle is indicated. For proper measuring of the side clearance angle at drill corner, this corner should be above the drill axis by distance \( a_o \) and both boundaries defining this angle should lie in the same working plane through drill corner.

Standard DIN 1414-2 provides methodology for measuring the clearance angle as follows: The side clearance angle \( \alpha_f \) shall be measured at the edge of one of the lands using a V-block with stop and a TMM. Place the drill in the V-block so that it is in contact with the stop. Rotate the drill until its axis lies approximately at the land edge (the major cutting edge will appear as a straight line parallel to the datum plane). Fix the drill in this position. Adjust the reticle so that it is over the land edge and take the reading. Rotate the drill through 180° and repeat this procedure.

It can be seen that the definition and tool location according to standard DIN 1414-2 are not the same as those defined by its twin standard DIN 1414-1:2006-11 in terms of the location of the major cutting edge and the boundaries of the clearance angle. Moreover, there are a number of problems with this methodology:

1. No particular system of drill geometry measurement is indicated although only the clearance angle in T-hand-S can be measured with a TMM.
2. In T-hand-S, the side clearance angle should be represented as shown in Figure 4.55, that is, the drill corner should be above the drill axis by distance \( a_o \) (Figure 4.66).
3. The term datum plane is not defined by the standard. The T-hand-S reference plane is actually meant.

![FIGURE 7.87](Image)

Clearance angle interpretations according standards DIN 1414-1 and DIN 1414-2.
Standard ASME B94.11M-1993 does not provide neither the methodology of the clearance angle measurement nor the tolerances on this angle. However, in USCTI book (The Metal Cutting Tool Institute 1989), which is the explanations to this standard, the clearance angle is depicted in the manner shown in Figure 7.88 and defined as the angle, measured across the margin, at the periphery of the drill. The position of the cutting edge shown in Figure 7.88 is not defined and it is not the same as defined by standard DIN 1414-1 and DIN 1414-2.

In the author’s opinion, the definitions of the side clearance angle by both standards are misleading. For example, the drill shown in Figure 7.87 (as represented in according to DIN 1414-2) has negative clearance angle as the top of its heels is ahead of the drill corners in the axial direction so that these tops will make the first contact with the workpiece (instead of drill corners) causing interference of the flank faces. In Figure 7.88, the lower boundary that defines this angle should be drawn from the drill corners not from the end of the heel as shown in the picture because the line of intersection of the flank face with the drill body is not a straight line. The upper boundary that defines this angle in Figure 7.88 is drawn from the top of the chisel edge, which is in direct contradiction with the standard (ISO 3002) definition of the clearance angle. Moreover, the plane of this angle measurement is unanswered as two boundaries shown in Figure 7.79 do not belong to the same plane.

To understand the major issue when defining and measuring the clearance angle, one should realize that any angle in the tool geometry consideration is measured in a certain reference plane as discussed in Appendix C and Chapter 4, that is, the boundaries of any angle should be in the same plane of measurement. Moreover, any inspection is a comparison of what was actually made to that set by the drawing. As the tool drawing sets the clearance angle at drill corner in T-hand-S (see example in Chapter 4, Figure 4.54), then it should be measured in the same system to make a meaningful comparison. Figure 7.89 shows the proper tool location in a TMM for the side clearance angle measurement at the drill corner A. As can be seen, the major cutting edge AB is set parallel to the drill longitudinal axis and shifted by distance \( a_o \) (see Chapter 4). Line 1 passing through drill corner A is the first boundary of the clearance angle that is set at 90° to the drill longitudinal axis. It represents the tangent to the assumed machining surface at point A. Line 2 passing through drill corner A is the second boundary of the clearance angle that is tangent to the flank face at point A. Note that both boundaries defining the side clearance angle at the drill corner lie in the same working plane through this corner. Under this setting, the side clearance angle at the drill corner is measured between the normals to line 1 and line 2 as shown in Figure 7.89. Although such setting for clearance angle determination at point A is commonly recommended by various standards and manuals, the problem with proper determining of the tangent to the curved flank at point A, particularly for small clearance angles and small drill diameters, is important though not clearly understood. Therefore, a need is felt to clarify this long-standing problem.

When the primary flank face is flat, there is no problem with defining the position of line 2 as the line of intersection of this flank face and the drill margin or cylindrical body is almost a
straight line when viewed in the manner shown in Figure 7.89. In this simple yet becoming common case, line 2 is just set along this line of intersection so that the side clearance angle $\alpha_f$ can be measured accurately. The problem is that this angle is not a part of the tool drawing for drills with the primary flat flanks because the normal clearance angle is indicated in the tool drawing for such drills as discussed in Chapter 4.

When the primary flank is not flat, the curve of intersection of the flank face and the drill margin and/or cylindrical body should be considered and characterized properly. In general, a curve is characterized by the curvature, radius of curvature, and center of curvature (Quadrini et al. 2007). They are a measure of how much or how quickly a line curves compared to a straight line. Figure 7.90 visualizes the basic terms involved. In the $x$–$y$ coordinate system shown in this figure, any point on this curve is characterized by its slope or tangent shown for point A in Figure 7.90. The rate of change of this tangent inclination with respect to the $x$-axis along the curve is called the curvature.
The center of curvature at considered point A is O. It is defined as the center of the circle whose center lies on the concave side of a curve on the normal to the tangent at a given point of the curve and whose radius $\rho_A$ is equal to the radius of curvature at that point.

The next step is to apply these properties of a curve to measuring the clearance angle. The proper determination of the clearance angles in various section planes for a drill with quadratic (Figure 4.61) and helical (known as CAM-relieved in the industry) point drills requires the assessment of the parameters of the curves formed as the intersection line of the sculptured flank face and the corresponding reference plane of the clearance angle measurement in T-hand-S. To determine the side clearance angle $\alpha_f$ (the clearance angle in the working plane in T-hand-S) at a point A of the major cutting edge, the working plane $P_f$ through point A should first be defined. As shown in Figure 7.91, it is defined by two lines: one is the tangent to the assumed bottom of the hole being drilled—it is the vertical line through point A parallel to the datum axis. Another is a line through point A drawn parallel to the axis of the datum. It is actually the normal to the tangent to the assumed bottom of the hole being drilled at point A.

The intersection line of the flank surface and the defined working plane is a curve for which parameters must be evaluated to measure the side clearance angle $\alpha_f$. According to the definition given in Chapter 4, the side clearance angle is the angle between two tangents: the tangent to the assumed bottom of the hole being drilled and the tangent to the curve representing the flank surface in the working plane. Figure 7.91a shows the case where the center of curvature O of this curve lies on the line through point A parallel to the datum axis. As such, the tangent to this curve at point A coincides with that to the assumed bottom of the hole being drilled. By definition, the side clearance angle in this case is equal to zero.

The case shown in Figure 7.91a presents a significant problem in tool inspection with any, even the most advanced, measuring equipment. For example, when one tries to follow the procedure presented in standard DIN 1414-2 (Figure 7.89) to measure a small side clearance angle (6°–8°), it is difficult to establish the position of line 2 (Figure 7.89) particularly for small drill diameters even when a modern TMM is used. Even the most advanced tool inspection machines measure this angle by measuring the distances between the tool flank curve and the tangent to the assumed bottom of the hole being drilled represented by a straight line through point A so that rather misleading results are common in such measurements. Moreover, visually, the flank surface appears as the proper grind as per Figure 7.88.
Figure 7.91b shows the proper location of the center of curvature $O_1$ of the curve representing the flank surface in the working plane. It has to be above the normal of the tangent to the assumed bottom of the hole being drilled at point A. As such, the side clearance angle at point A is determined as

$$\alpha_{fA} = \arcsin \frac{OO_1}{\rho_A}$$

where $\rho_A$ is the radius of curvature of the flank face at point A.

No matter what type of nonflat flank surface grinding is used, the meaning of distance $OO_1$ is the same and it is always determined by Equation (7.14).

### 7.7.13 Surface Roughness

Standard ASME B94.11M-1993 does not specify surface roughness on drills.

Standard GOST 2034-80 specifies the following parameters of drill surfaces roughness:

- Flank faces: for high-precision drills A1 and for drills B1, $Ra = Rz = 3.2 \, \mu m$; for drills B, $Rz = 6.3 \, \mu m$.
- For margins: for high-precision drill A1, $Ra = Rz = 3.2 \, \mu m$; for drills B1 and B, $Rz = 6.3 \, \mu m$.
- Flute face: at the drill point, $Rz = 3.2 \, \mu m$ and then $Rz = 6.3 \, \mu m$.
- Shank: for high-precision drills, $Ra = 0.63 \, \mu m$; for drills of normal precision, $Ra = 1.25 \, \mu m$.

Standard DIN 1414-1:2006-11 specifies surface roughness for various drill elements as shown in Table 7.34. Standard DIN 1414-2 specifies the following places for measuring surface roughness of the drills:

The arithmetical mean deviation of the profile $Ra$ shall be measured using a suitable stylus instrument at the following points:

a. In the flute, at right angles to the helix angle, $\omega_d$, at a distance equal to the drill diameter from the point of the drill (measurement 1), and at half the flute length (measurement 2)

b. At the point, across the grinding direction close to the major cutting edges
c. Across the land, between the flute and the end of the relief
d. On the Morse taper, close to point A as in DIN 228-1 (the gage line diameter)

When measuring in the flute (item a), the length of measurement shall extend to the minor cutting edge (the margin). Measurement across the land (item c) shall be made along the drill axis in either direction.

In the author’s opinion, the roughness of drill’s surfaces according to standards GOST 2034-80 and DIN 1414-1:2006-11 is course for HP drills. Figure 7.92 shows the recommended surface roughness for various drill elements as shown in Table 7.34.

**Table 7.34**

<table>
<thead>
<tr>
<th>Drill Element</th>
<th>$\leq 15 , mm$</th>
<th>$&gt; 15 , mm$</th>
</tr>
</thead>
<tbody>
<tr>
<td>Land (margin)</td>
<td>$Ra = 0.8 , \mu m$</td>
<td>$Ra = 0.8 , \mu m$</td>
</tr>
<tr>
<td>Shank</td>
<td>$Ra = 0.8 , \mu m$</td>
<td>$Ra = 0.8 , \mu m$</td>
</tr>
<tr>
<td>Flute</td>
<td>$Ra = 0.8 , \mu m$</td>
<td>$Ra = 1.6 , \mu m$</td>
</tr>
<tr>
<td>Flute at drill point</td>
<td>$Ra = 0.8 , \mu m$</td>
<td>$Ra = 0.8 , \mu m$</td>
</tr>
</tbody>
</table>
Surface roughness (not worse than) is assigned for any component of the drill point and the flute. The standard for its assessment is clearly indicated.

- The direction of the surface roughness measurement is clearly indicated on each drill component.

- The surface roughness on the should be always assigned. The axis of the shank is the datum so the shank should be made almost perfect in order to establish this axis properly by a high-precision tool holder. To do this, (a) the shank is ground to mirror-shining surface finish \( Ra = 0.3 \, \mu m \), and the mixed direction of grinding marks is assigned. The latter is achieved when most modern grinding machines having the second compensating grinding wheel are used to grind the shank and (b) 2.5 \( \mu m \) roundness tolerance is assigned, that is, the shank roundness should not be worse than 2.5 \( \mu m \) at any diameter along its length. Tolerance h6 is assigned on the shank diameters, which is standard for hydraulic and shrink-fit precision tool holders.

- The place for toolmarking is clearly indicated, and it is not on the shank.

- The flute washout radius, the flute length, and the minimum regrind flute length are shown.

The author clearly understands that there are only very few drill manufacturers in the world capable of achieving and properly inspecting such characteristics of the drill components as state-of-the art production and inspection equipment in addition to well-trained personnel (operators and engineers) are required. The testing and implementation of the drills made to these requirements proved their high efficiency in highly demanding automotive applications.
7.7.14 Drill Inspection

Any, even the simplest, inspection is a part of a quality program, so the terms involved should be clearly defined. This section defines the objective and basic terms of drill inspection to help both the drill manufacturers and the drill users to achieve HP drilling operations:

**Article**—an article is defined as a design or geometrical component of a drill. The article exists if and only if a characteristic(s) of this article is (are) assigned by the drill drawing or by other related document(s) properly referenced in the drill drawing.

**Characteristic**—a dimensional, visual, functional, mechanical, or material feature or property, which describes and constitutes the design of an article and can be measured, inspected, tested, or verified to determine conformance to design requirements. This includes drawing requirements (including requirements set in the drawing title block) and/or specification requirements determined from drawing notes. Note that dimensions/features identified as Reference (i.e., noted as REF or are in parenthesis in the drawing) are exempt and are not required to be inspected/reported.

**First Article Inspection (FAI)**—a complete verification that the article being inspected complies with the requirements identified in engineering drawings, specifications, and purchase orders as well as any other applicable design requirement(s). The FAI package includes the FAI form, annotated drawing, and all supporting documentation (i.e., certifications test reports). Note that the FAI can also be referred to as the First Article Inspection Report (FAIR) or the Initial Sample Inspection Report (ISIR).

**Defect**—a defect is defined as a nonconforming characteristic.

**Defective drill**—a drill that contains one or more defects.

**Lot**—a collection of drills bearing identification and treated as a unique entity from which a sample is to be drawn and inspected to determine conformance with the acceptability criteria.

**Homogeneous lot**—a group of drills manufactured at approximately the same time that are expected to share similar quality levels for selected characteristics.

**Lot size**—the number of drills in a lot.

**Random sample**—a sample selected in such a way that each drill of the population has an equal chance of being selected.

**Reject**—to refuse to accept. Rejection in an acceptance sampling sense means to decide that a lot has not been shown to satisfy the acceptance criteria based on the information obtained from the sample. Rejected lots may be immediately submitted for material review disposition upon completion of sampling inspection or may be screened 100% for any detected defects. In this case, all defective drills should be submitted for material review.

**Sample**—part of a population selected according to some rule or plan.

**Sample size**—the number of drills selected as representative of a population.

**Sampling plan**—the instructions given to personnel responsible for performing sampling inspection. Sample inspection plans (as a part of Standard Operating Practice) should be developed, authorized for use, and used in drill inspection and are defined in attachments to this.

FAI is required whenever any of the following occur:

- The tools are being manufactured for the first time by this manufacturer
- There is a revision to the tool engineering drawing
- There is a change in raw material used (use of an alternate material listed in the drawing)
- There is a change in tooling (new, replacement, or major modification) used in drill manufacturing
- There is a change in the manufacturing process that may affect form, fit, or function of the part
- The drill has not been manufactured in 6 months
- There is a change in the plant of manufacture (either due to a natural or man-made event)
- There is a change in the supplier's sub-tier special process provider (i.e., heat-treat, coating/coating facility)
- There is a customer requirement for FAI as a part of the corresponding QC procedure or when the tools in a particular manufacturing batch do not perform as expected, that is, have lower than usual tool life and break.

Table 7.35 presents an example of the process flow for a hypothetical VPDrills company. The following should be noted in this table:

1. Quality inspection is a serious procedure that must be carried out according to the developed quality plans, procedure, instructions, and manual to assure the high quality and sustainability of manufactured drills.
2. The inspection including FAI is based exclusively on the tool drawing that must include all the information essential to tool performance and manufacturing. In other words, if an article is not properly specified by the tool drawing, it cannot be inspected according to the basic definition of inspection.

The inspection includes the following:

1. **Visual inspection** carried out according to the note(s) in the tool drawing. For example, a note in a tool drawing “3. CUTTING TIP AND ITS EDGES SHOULD NOT HAVE ANY VISIBLE CHIPS, CRACKS OR DEFECTS @ x25” implies that the drill should be cleaned before visual inspection that is carried out under a magnification glass/microscope having magnification ×25. Another example is the location and authenticity of drill marking (e.g., *laser etch location* in Figure 7.92). Any visual inspection relies on the experience and judgment of the inspector.
2. **Dimensional inspection** includes inspection of the tool dimensional and geometrical articles. Dimensional inspection planning is an activity to generate specific instructions to inspect manufactured drills based on the design. Properly developed inspection plans ensure consistency of measurement results. Inspection planning activity and data models are necessary to enable inspection planners and product designers to effectively communicate during product design and inspection process planning.
3. **Physical inspection** includes the inspection of physical characteristics of the article assigned by the drawing, for example, hardness, thickness, and consistency of the coating layer, and MWF flow rate through the coolant holes of the tool under a specified inlet MWF pressure. Physical inspection can be either nondestructive or destructive, for example, when the strength of brazing of the cutting insert and the tool body is evaluated.

The visual inspection is carried out according to the note in the tool drawing or according to the company’s quality procedure/manual for such an inspection. The physical inspection characteristics and procedures as related to the tool material(s) are discussed in Chapter 3. Therefore, the section to follow concentrates on the dimensional inspection the tool dimensional and geometrical articles as this type of inspection is the most common yet least understood in terms of geometrical articles including their proper definition and tolerances.

### 7.7.15 Dimensional Inspection (Metrological) System: Flowchart

The dimension inspection is a part of the dimension metrology system. Its flow chart shown in Figure 7.93 includes four principal stages: design, planning, execution, and analysis. According to the author’s experience, the first stage presents the major problem in drill inspection.
<table>
<thead>
<tr>
<th>Major Steps</th>
<th>Activities</th>
</tr>
</thead>
<tbody>
<tr>
<td>01 FAI sample selection</td>
<td>After all modifications to methods of manufacture have been made, a drill from the first production lot shall be chosen for FAI.</td>
</tr>
<tr>
<td>02 Drawing selection</td>
<td>1. Internal FAI: Use the manufacturing shop copy (SC) drawing unless otherwise specified on the inspection plan of VPDrills.</td>
</tr>
<tr>
<td></td>
<td>2. Customer FAI: Use the customer print (CP) approved by the customer (if one exists). Otherwise use the customer copy (CC) if a CP is not available or SC as specified on the inspection plan.</td>
</tr>
<tr>
<td>03 Characteristic identification</td>
<td>1. Determine the FAI form from the database of VPDrills to be used to complete the FAI. Refer to the quality plan for customer-specific requirements for completing the FAI.</td>
</tr>
<tr>
<td></td>
<td>2. Obtain a hard copy of the engineering drawing and manually place consecutively numbered circles (balloons) adjacent to each characteristic in the drawing. As an alternate, a CAD program may (e.g., Inventor) be used to identify the Inventor characteristics in the drawing.</td>
</tr>
<tr>
<td></td>
<td>3. Record each characteristic in the appropriate location on the FAI form corresponding to the numbered circle in the drawing. For example, the first circled characteristic in the drawing would be recorded in the row with item no. 1 identified. Subsequent rows are added consecutively until all characteristics are identified. Reference characteristics may be omitted in the FAI form.</td>
</tr>
<tr>
<td>04 First Article Inspection</td>
<td>1. Unless otherwise approved by VPDrills quality, all inspection methods must be in accordance with VPDrills specification VPA-1203-2012 Standard Measuring Methods and stated for each characteristic on the form. Units of measure are those that are specified in the VPDrills engineering drawing.</td>
</tr>
<tr>
<td></td>
<td>2. Record the variable measurement of all characteristics in the Actual Measure block (or equivalent) of the row corresponding to the item number noted in the drawing:</td>
</tr>
<tr>
<td></td>
<td>a. All dimensional, hardness, and test requirements must be reported with a measurement value.</td>
</tr>
<tr>
<td></td>
<td>b. Dimensions stating TYP or multiple locations (e.g., 2x, 4x) must report measurement values for each feature.</td>
</tr>
<tr>
<td></td>
<td>c. All basic dimensions must be reported (unless exclusion is approved by VPDrills quality).</td>
</tr>
<tr>
<td></td>
<td>d. Geometrical features (i.e., true position) for multiple features must report the condition for each feature.</td>
</tr>
<tr>
<td></td>
<td>3. For characteristics that are recorded on material and/or test report certifications, type CONFORMS in the Actual Measure block (or equivalent), reference the certification number in the Comment block (or equivalent), and include a copy of the certification(s) with the FAI submittal.</td>
</tr>
<tr>
<td></td>
<td>4. For characteristics that require visual inspection, type CONFORMS in the Actual Measure block (or equivalent) and state VISUAL in the Inspection Method block (or equivalent).</td>
</tr>
<tr>
<td></td>
<td>5. For characteristics that are inspected using a functional or go/no-go gage, type CONFORMS in the Actual Measure block (or equivalent) and state the gage type or gage number in the Inspection Method block (or equivalent).</td>
</tr>
<tr>
<td></td>
<td>6. Indicate acceptance or rejection by typing A or R in the block provided.</td>
</tr>
<tr>
<td></td>
<td>7. Sign and date the form in the appropriate boxes. Note: an electronic signature or typed signature is acceptable.</td>
</tr>
</tbody>
</table>

(continued)
While dimensional articles and their characteristics indicated in the proper tool drawing are clear (at least, most of the time), geometrical articles is of prime concern as their definitions, designation in tool drawings, and tolerancing vary from one drill drawing to the next, from one drill manufactures to the other. This is due to coexistence of many contradictive sources of information on the matter including national and international standards as well as books including textbooks and web-based materials. Therefore, the subsequent section aims to provide methodological help to specialist to carry out this stage.
7.7.16 Dimensional Inspection (Metrological) System: Design Stage

As can be seen in Figure 7.93, the dimension inspection system originates from the tool drawing, so it completely relies on the information (and its accuracy) about articles provided by this drawing. Provided that one has a proper drill drawing where all information essential to the tool manufacturing and performance is indicated properly, he or she is fully equipped to go to the second design step (Figure 7.93) that includes compiling a list of articles and their characteristics. Tables 7.36 and 7.37 list articles to be considered and provide their proper definitions with references to the corresponding graphical representations. Although these tables give definitions for the listed articles, some additional explanations are required to the additional terms used in the article definitions, feasibility of their measurement, and adjustments to particularities of various drill designs.

7.7.16.1 Diameter-/Length-Related Articles

*Back taper* is defined as article 07N in Table 7.36. This article characteristic (designated as $\Delta_{bt}$ in Chapter 4 [Section 4.5.9.5]) is assigned in the tool drawing as the difference in the diameters over a specified reference length $L_{bt-r}$ (see Figure 7.94), that is,

$$\Delta_{bt} = d_{b1} - d_{b2}$$  \hspace{1cm} (7.15)

The first diameter $d_{b1}$ is normally the drill diameter and the second $d_{b2}$ is measured according to the standard definition at the reference distance (length) $L_{bt-r}$ from the first as shown in Figure 7.94. For example, standard DIN 1414-1 recommends the back taper to be in the range of 0.02–0.08 mm/100 mm of drill length, that is, the reference length is 100 mm as discussed in Section 7.7.9. This rather great reference length $L_{bt-r}$ creates two problems:

- The overall drill length can be shorter than length $L_{bt-r}$ or the length of the special drill material (e.g., PCD) can be much shorter than this length
- Back taper is normally applied along some distance from the drill point, and thus, the rest of the drill is made cylindrical.

In both cases, back taper cannot be physically measured over the reference length $L_{bt-r}$.

To solve the problem with back taper assessment, the second diameter $d_{b2-a}$ is measured at some known feasible distance $L_{bt-a}$ from the drill as shown in Figure 7.94. Using geometric similarity as

$$\frac{d_{b1} - d_{b2}}{L_{bt-r}} = \frac{d_{b1} - d_{b2-a}}{L_{bt-a}}$$  \hspace{1cm} (7.16)

one can determine the back taper as

$$\Delta_{bt} = (d_{b1} - d_{b2-a}) \frac{L_{bt-r}}{L_{bt-a}}$$  \hspace{1cm} (7.17)

Margin width, $b_{m}$, is defined as article 08N; body clearance diameter, $d_{cl}$, as article 09N; and land width, $W_{L}$, as article 010N in Table 7.36. The characteristics of these articles are measured in the back plane as shown in Figure 7.95.

Coolant hole diameter, $d_{ch}$, is defined as article 14N in Table 7.36. It is measured in the back plane. Figure 7.95 shows that two diameters are measured in the case of two-hole drill, that is, $d_{ch-1}$ and $d_{ch-2}$. It is obvious that if there are more than two coolant holes, the diameter of each hole is measured.

The diameter of the auxiliary margins is defined as article 15N in Table 7.36. Figure 7.96 shows the drill designs where the diameter of the auxiliary margins, $d_{m-a}$, is not the same as that of the drill. Figure 7.96a shows drill design discussed in Chapter 4 (Figure 4.165). Such a design includes burningishing margins, which have a diameter that is slightly greater than that of the drill to improve drill stability and achieve greater quality of the machined surface. Figure 7.96b shows a PCD-tipped drill. The diameter of the auxiliary margins is slightly smaller than that of the drill (normally 6–12 μm).
### TABLE 7.36
List and Definitions of the Non-Datum Articles

<table>
<thead>
<tr>
<th>#</th>
<th>Article</th>
<th>Definition</th>
<th>Graphical Representation</th>
</tr>
</thead>
<tbody>
<tr>
<td>01N</td>
<td>Drill diameter</td>
<td>The diameter over the margins of the drill measured at the periphery corners. If a drill has several steps, the diameter of each step of the drill is measured over the margins at the corresponding periphery corners.</td>
<td>Size (d_{o}) in Figure 7.50. Sizes (d_1) and (d_2) in Figure 7.46.</td>
</tr>
<tr>
<td>02N</td>
<td>Overall length</td>
<td>The length from the drill point to the end of its shank.</td>
<td>Size (L_{oa}) in Figure 7.46.</td>
</tr>
<tr>
<td>03N</td>
<td>Flute length</td>
<td>The length from the drill point to the end of the flute.</td>
<td>Size (L_{f}) in Figure 7.46.</td>
</tr>
<tr>
<td>04N</td>
<td>Active flute length</td>
<td>The length from the drill point to the place of the flute where it begins to change its profile.</td>
<td>Size (L_{f,0}) in Figure 7.50.</td>
</tr>
<tr>
<td>05N</td>
<td>Length of the body relief</td>
<td>The length from the drill point to the end of the body relief.</td>
<td>Size (L_{br}) in Figure 7.50.</td>
</tr>
<tr>
<td>06N</td>
<td>Step length</td>
<td>The difference in diameters over the margins of the drill per unit (specified) length of the drill.</td>
<td>Size (L_1) Figure 7.46.</td>
</tr>
<tr>
<td>07N</td>
<td>Back taper</td>
<td>Margin is the cylindrical portion of the land that is not cut away to provide clearance. The width of the margin is measured in the back (perpendicular to the drill longitudinal axis) plane.</td>
<td>See Figure 7.94.</td>
</tr>
<tr>
<td>08N</td>
<td>Margin width</td>
<td>Margin is the cylindrical portion of the land that is not cut away to provide clearance. The width of the margin is measured in the back (perpendicular to the drill longitudinal axis) plane.</td>
<td>See Figure 7.95.</td>
</tr>
<tr>
<td>09N</td>
<td>Body clearance diameter</td>
<td>The diameter of the body clearance cylinder, that is, the portion of the land that has been cut away to prevent its rubbing against the walls of the hole being drilled. This diameter is measured in the back (perpendicular to the drill longitudinal axis) plane.</td>
<td>See Figure 7.95.</td>
</tr>
<tr>
<td>10N</td>
<td>Land width</td>
<td>The distance between the leading edge and the heel of the land measured at right angles to the leading edge.</td>
<td>See Figure 7.95.</td>
</tr>
<tr>
<td>11N</td>
<td>Shank diameter (if cylindrical)</td>
<td>Shank is the part of the drill by which it is held and driven.</td>
<td>See Figure 7.53.</td>
</tr>
<tr>
<td>12N</td>
<td>Shank out of roundness (circularity)</td>
<td>Out of roundness is defined as a half of the maximum difference in the diameters of two concentric circles within which each circular element of the shank surface is located. It is measured in a plane perpendicular to the shank longitudinal axis.</td>
<td>See definition in Figure 7.30 and designation in a tool drawing in Figure 7.92.</td>
</tr>
<tr>
<td>13N</td>
<td>Shank length</td>
<td>The distance between two ends of the shank in the axial direction.</td>
<td>See Figure 7.92.</td>
</tr>
<tr>
<td>14N</td>
<td>Coolant hole diameter</td>
<td>A coolant hole is the passage through which the MWF is fed through the drill body (for drills with internal MWF supply). The diameter of each coolant hole is measured and reported. The coolant hole diameters is measured in the back (perpendicular to the drill longitudinal axis) plane.</td>
<td>See Figure 7.95.</td>
</tr>
</tbody>
</table>
This is because these margins are used only for improving drill entrance stability, particularly in drilling cored holes. When the drill is fully engaged in cutting, these margins should not touch the surface of the machined hole by PCD tips. In both cases, the diameter of the auxiliary margins, $d_{m-a}$, is of critical importance for drill performance so it should be tightly toleranced and thus inspected.

Flute profile is defined as article 24N in Table 7.36. Although the tool profile can be controlled by measuring the boundary curves and their radii of curvature, it is not feasible for many drills having complicated flute profiles. The easiest way to inspect the accuracy of the actual tool profile is to import DXF overlays (2D CAD outline curves) into the measurement view.

### 7.7.16.2 Major Reference Plane

The characteristics of many important drill articles are measured in or with respect to the major reference plane that should be first distinguished among infinite numbers of reference planes containing the axis of the datum (commonly, the axis of the shank) as was introduced in Chapter 4 (Section 4.5.5). The major reference plane is defined as the plane containing the axis of the datum and as follows:

1. In the sense of the reference plane discussed in Appendix C, it should be perpendicular to the assumed direction of the cutting speed, which in T-hand-S is always perpendicular to the cutting edge. In geometrical sense, it is parallel to the projections of the major cutting edges into the back plane when the major cutting edges are 2D straight or curved lines as shown in Figure 7.97a.

2. The plane of radial symmetry at 180° for other shapes of the major cutting edges. Figure 7.98 shows an example for CoroDrill 860 drill (Sandvik Coromant Co.).
### TABLE 7.37
List and Definitions of the Datum-Related Dimensional Articles

<table>
<thead>
<tr>
<th>#</th>
<th>Article</th>
<th>Definition</th>
<th>Graphical Representation</th>
</tr>
</thead>
<tbody>
<tr>
<td>01D</td>
<td>Drill runout</td>
<td>The distance between the datum axis and the center of the margin circle measured in the back (perpendicular to the datum axis) plane through the drill corners.</td>
<td>See Figure 7.59.</td>
</tr>
<tr>
<td>02D</td>
<td>Point angle, $\Phi_p$</td>
<td>Measured in the major reference plane as the angle between the projections of the major cutting edges (lips) into this plane. When the projections of the major cutting edges into the major reference plane are not straight lines, the point angle $\Phi_{p,i}$ is measured for a point of consideration $i$ on this projections as shown in Figure 7.96c. When the drill point is made with more than one point angle (e.g., a triple-point drill shown in Figure 4.158), the point angle is measured for each part.</td>
<td>See Figure 7.97.</td>
</tr>
<tr>
<td>03D</td>
<td>Half-point angle, $\phi$</td>
<td>It is measured individually for each projection of the major cutting edges as the angle between the projection of a considered major cutting edge (lip) into the major reference plane and the axis of the datum as shown in Figure 7.96b. In the case of a two-flute drill considered in Figure 7.96b, two half-point angles, $\phi_1$ and $\phi_2$, are measured. When the projections of the major cutting edges into the major reference plane are not straight lines, the half-point angle $\phi$ is measured for a point of consideration $i$ on this projection of the corresponding major cutting edge. In the case of a two-flute drill shown in Figure 7.96c, two half-point angles, $\phi_{1,i}$ and $\phi_{2,i}$, are measured. When the drill point is made with more than one point angle (e.g., a triple-point drill shown in Figure 4.158), the half-point angles are measured for each part of the considered cutting edge.</td>
<td>See Figure 7.97.</td>
</tr>
<tr>
<td>04D</td>
<td>Flute spacing, $a_o$</td>
<td>It is measured in the back plane. It is defined as the distance between the projection of the cutting edge in the back plane and the line of intersection of the major reference plane and the back plane. See explanations in Section 7.7.16.3.</td>
<td>Figures 7.97a and 7.99</td>
</tr>
<tr>
<td>05D</td>
<td>Relative lip height (lip height variation), $\Delta_{lh}$</td>
<td>Measured in the major reference plane, the normal (relative lip height) or the axial (parallel to the datum axis and known as the axial runout) distance between the projections of two or more cutting edges into the major reference plane conditionally revolved about the axis of the datum to bring all the edges on the same side of the major reference plane with respect to the axis of the datum. In the case of two-flute drill, one projection of the major cutting edge is revolved by 180°. See explanations in Section 7.7.16.3.</td>
<td>Figure 7.100</td>
</tr>
</tbody>
</table>
### TABLE 7.37 (continued)
List and Definitions of the Datum-Related Dimensional Articles

<table>
<thead>
<tr>
<th>#</th>
<th>Article</th>
<th>Definition</th>
<th>Graphical Representation</th>
</tr>
</thead>
<tbody>
<tr>
<td>06D</td>
<td>Normal rake angle</td>
<td>The angle between rake face and the reference plane $P_r$ measured in the normal plane $P_n$. If the rake face surface is not planar, then the plane tangent to the curved rake face surface in the considered point on the cutting edge is used instead of the rake plane. See explanations in Section 7.7.16.3.</td>
<td>Figure 7.101</td>
</tr>
<tr>
<td>07D</td>
<td>Side rake angle</td>
<td>The angle between rake face and the reference plane $P_r$ measured in the working plane $P_f$. If the rake face surface is not planar, then the plane tangent to the curved rake face surface in the considered point on the cutting edge is used instead of the rake plane. See explanations in Section 7.7.16.3.</td>
<td>Figure 7.101</td>
</tr>
<tr>
<td>08D</td>
<td>Back rake angle</td>
<td>The angle between rake face and the reference plane $P_r$ measured in the back plane $P_p$. If the rake face surface is not planar, then the plane tangent to the curved rake face surface in the considered point on the cutting edge is used instead of the rake plane.</td>
<td>Figure 7.101</td>
</tr>
<tr>
<td>09D</td>
<td>Normal clearance angle</td>
<td>The angle between the tool cutting edge plane $P_s$ and the tool primary flank plane measured in the normal plane $P_n$. It is clear that if the primary flank surface is not planar, then the plane tangent to the curved flank surface in the considered point on the cutting edge is used instead of the flank plane. See explanations in Section 7.7.16.3.</td>
<td>Figure 7.101</td>
</tr>
<tr>
<td>10D</td>
<td>Side clearance angle</td>
<td>The angle between the tool cutting edge plane $P_s$ and the tool primary flank plane measured in the working plane $P_f$. It is clear that if the primary flank surface is not planar, then the plane tangent to the curved flank surface in the considered point on the cutting edge is used instead of the flank plane. See explanations in Section 7.7.16.3.</td>
<td>Figure 7.101</td>
</tr>
<tr>
<td>11D</td>
<td>Back clearance angle</td>
<td>The angle between the tool cutting edge plane $P_s$ and the tool primary flank plane measured in the back plane $P_p$. It is clear that if the primary flank surface is not planar, then the plane tangent to the curved flank surface in the considered point on the cutting edge is used instead of the flank plane. See explanations in Section 7.7.16.3.</td>
<td>Figure 7.101</td>
</tr>
<tr>
<td>12D</td>
<td>Normal clearance angle</td>
<td>The angle between the tool cutting edge plane $P_s$ and the tool secondary flank plane measured in the normal plane $P_n$. It is clear that if the secondary flank surface is not planar, then the plane tangent to the curved flank surface in the considered point on the cutting edge is used instead of the flank plane. See explanations in Section 7.7.16.3.</td>
<td>Figure 7.101</td>
</tr>
</tbody>
</table>
### TABLE 7.37 (continued)
List and Definitions of the Datum-Related Dimensional Articles

<table>
<thead>
<tr>
<th>#</th>
<th>Article</th>
<th>Definition</th>
<th>Graphical Representation</th>
</tr>
</thead>
<tbody>
<tr>
<td>13D</td>
<td>Side clearance angle (secondary flank)</td>
<td>The angle between the tool cutting edge plane $P_s$ and the tool secondary flank plane measured in the working plane $P_f$. It is clear that if the secondary flank surface is not planar, then the plane tangent to the curved flank surface in the considered point on the cutting edge is used instead of the flank plane. See explanations in Section 7.7.16.3.</td>
<td>Figure 7.102</td>
</tr>
<tr>
<td>14D</td>
<td>Back clearance angle (secondary flank)</td>
<td>The angle between the tool cutting edge plane $P_s$ and the tool secondary flank plane measured in the back plane $P_p$. It is clear that if the secondary flank surface is not planar, then the plane tangent to the curved flank surface in the considered point on the cutting edge is used instead of the flank plane. See explanations in Section 7.7.16.3.</td>
<td>Figure 7.101</td>
</tr>
<tr>
<td>15D</td>
<td>Width of primary clearance, $b_{fn}$</td>
<td>Length of the projection of the primary clearance surface into the tool cutting edge plane $P_s$ measured in the normal plane $P_n$.</td>
<td>Figure 7.101</td>
</tr>
<tr>
<td>16D</td>
<td>Edge preparation</td>
<td>A transition curve between the rake and the flank surfaces measured in the T-hand-S normal plane (direction normal to the theoretical cutting edge). The radius of the cutting edge, $R_{ce}$, that approximates the transition curve between the rake and flank faces is measured in the normal plane. See explanations in Section 7.7.16.3.</td>
<td>Figure 7.101</td>
</tr>
</tbody>
</table>

**Articles related to the chisel edge and its region**

<table>
<thead>
<tr>
<th>#</th>
<th>Article</th>
<th>Definition</th>
<th>Graphical Representation</th>
</tr>
</thead>
<tbody>
<tr>
<td>17D</td>
<td>Chisel angle, $\psi_{cl}$</td>
<td>The angle between the projection of the chisel edge in the back plane and the major reference plane measured in the back plane.</td>
<td>Figure 7.103a</td>
</tr>
<tr>
<td>18D</td>
<td>Chisel edge length, $l_{cl}$</td>
<td>The length of the projection of the chisel edge into the back plane.</td>
<td>Figure 7.103a</td>
</tr>
<tr>
<td>19D</td>
<td>Chisel edge centrality (symmetry), $\Delta_c$</td>
<td>The distance between the projection of the chisel edge into the back plane and the axis of the datum.</td>
<td>Figure 7.103b</td>
</tr>
<tr>
<td>20D</td>
<td>Centrality of the web, $\Delta_w$</td>
<td>The difference of the distances from the axis of the datum and the cutting edges measured along a line drawn through the point of the intersection of the axis of the datum with the back plane perpendicular to the major reference plane. See explanations in Section 7.7.16.4.</td>
<td>Figure 7.103c</td>
</tr>
<tr>
<td>21D</td>
<td>Web thickness, $d_{wb}$</td>
<td>The distance between the points of the major cutting edges measured along a line drawn through the point of the intersection of the axis of the datum with the back plane perpendicular to the major reference plane. See explanations in Section 7.7.16.4.</td>
<td>Figure 7.103c</td>
</tr>
<tr>
<td>22D</td>
<td>Radius of the end of the chisel edge, $r_{cl}$</td>
<td>Only for S chisel edge drills. The radius of the circular arc tangent to both the straight portions of the chisel edge and the corresponding major cutting edge. See explanations in Section 7.7.16.4.</td>
<td>Figure 7.104a</td>
</tr>
<tr>
<td>23D</td>
<td>Mismatch of the intersection lines of the secondary flank faces with the primary flank faces, $m_f$</td>
<td>The distance between the points of intersection of the lines of intersection of the primary and secondary flank faces with the chisel edge measured in the back plane. See explanations in Section 7.7.16.4.</td>
<td>Figure 7.104b</td>
</tr>
</tbody>
</table>
### TABLE 7.37 (continued)
List and Definitions of the Datum-Related Dimensional Articles

<table>
<thead>
<tr>
<th>#</th>
<th>Article</th>
<th>Definition</th>
<th>Graphical Representation</th>
</tr>
</thead>
<tbody>
<tr>
<td>24D</td>
<td>Chisel wedge angle, $\nu_{cl}$</td>
<td>Measured in the chisel edge reference plane as the angle between the projections of the rake and flank surfaces of the chisel into this plane.</td>
<td>Figure 7.105</td>
</tr>
<tr>
<td>25D</td>
<td>Chisel wedge symmetry, $\Delta_{clw}$</td>
<td>Only for self-centered designs of the chisel edge. The difference in the half chisel wedge angles, $\nu_{cl1}$ and $\nu_{cl2}$, measured in the chisel edge reference plane.</td>
<td>Figure 7.105b</td>
</tr>
<tr>
<td>26D</td>
<td>Chisel edge rake angle, $\gamma_{cl}$</td>
<td>The angle between the chisel edge rake face and the chisel edge reference plane measured in the chisel edge normal plane. If the rake face surface of the chisel edge is not planar, then the plane tangent to the curved rake face surface in the considered point on the cutting edge is used instead of the rake plane. See explanations in Section 7.7.16.4.</td>
<td>Figure 7.106</td>
</tr>
<tr>
<td>27D</td>
<td>Chisel edge clearance angle, $\alpha_{cl}$</td>
<td>The angle between the flank face of the chisel edge and the tool chisel edge plane, $P_{s-cl}$, measured in the chisel edge normal plane. If the flank face surface of the chisel edge is not planar, then the plane tangent to the curved flank face surface in the considered point on the cutting edge is used instead of the flank plane. See explanations in Section 7.7.16.4.</td>
<td>Figure 7.106</td>
</tr>
<tr>
<td>28D</td>
<td>Only for generic design of split-point drills.</td>
<td>Length of the rake face measured in the chisel edge normal plane. See explanations in Section 7.7.16.4.</td>
<td>Figure 7.107</td>
</tr>
<tr>
<td>29D</td>
<td>Only for generic design of split-point drills.</td>
<td>Transition radius, $r_{sp}$</td>
<td>Figure 7.107a</td>
</tr>
<tr>
<td></td>
<td>Gash-related parameters (generic)</td>
<td></td>
<td></td>
</tr>
<tr>
<td>30D</td>
<td>Gash face angle, $\psi_{gh-b}$</td>
<td>The angle between the sides of the gash measured in the back plane. See explanations in Section 7.7.16.5.</td>
<td>Figure 7.108</td>
</tr>
<tr>
<td>31D</td>
<td>Gash inclination angle, $\lambda_{gh-b}$</td>
<td>The angle between the straight and gashed portions of the major cutting edge measured in the back plane. See explanations in Section 7.7.16.5.</td>
<td>Figure 7.108</td>
</tr>
<tr>
<td>32D</td>
<td>Extent of the gash past the CL, $a_4$</td>
<td>Measured in the back plane, the distance between the end of the straight portion of the gash and the vertical axis of the drill drawn through the axis of the datum. See explanations in Section 7.7.16.5.</td>
<td>Figure 7.108</td>
</tr>
<tr>
<td>33D</td>
<td>Radius of the gash, $r_{gb}$</td>
<td>Radius of the gash measured in the back plane. See explanations in Section 7.7.16.5.</td>
<td>Figure 7.108</td>
</tr>
<tr>
<td>34D</td>
<td>Gash plane angle, $\psi_{gb-e}$</td>
<td>The angle between the gash direction and the axis of the datum measured in the major reference plane. See explanations in Section 7.7.16.5.</td>
<td>Figure 7.108</td>
</tr>
<tr>
<td></td>
<td>Articles related to the coolant holes</td>
<td></td>
<td></td>
</tr>
<tr>
<td>34D</td>
<td>Bolt diameter, $d_{ld}$</td>
<td>Diameter of a circle drawn from the axis of the datum and passing through the centers of the coolant holes. Measured in the back plane.</td>
<td>Figure 7.112</td>
</tr>
<tr>
<td>35B</td>
<td>Coolant holes location angle, $\psi_{cb}$</td>
<td>Angle between the line drawn through the centers of the coolant holes and the major reference plane measured in the back plane.</td>
<td>Figure 7.112</td>
</tr>
</tbody>
</table>
FIGURE 7.94 Back taper particularities: (a) for a twist drill and (b) for a straight-flute drill.

FIGURE 7.95 Showing graphical representations of margin width (article 08), body clearance diameter (article 09), land width (article 10), and coolant hole diameters (article 14) in Table 7.36.

FIGURE 7.96 Diameter of the auxiliary margins is not the same as that of the drill: (a) greater and (b) smaller.
Once the major reference plane is defined, the drill back plane can also be defined as the plane perpendicular to the major reference plane. The characteristics of the many important articles of the drill geometry are measured in the reference and back planes.

### 7.7.16.3 Articles Related to the Major Cutting Edges

The **point angle**, $\Phi_p$, is defined as article 02D in Table 7.37. The **half-point angle**, $\varphi$, is defined as article 03D in Table 7.37. **Flute spacing**, $a_o$, is defined as article 04D in Table 7.37. In the case of two-flute drill considered in Figure 7.97a, two heights, namely, $a_o{}_{AB}$ and $a_o{}_{CD}$, are measured as the flute spacing for the major cutting edges AB and CD, respectively, so that the absolute value of the difference $a_o{}_{AB} - a_o{}_{CD}$ is then compared with the tolerance $A_5$ (Section 7.7.7.3, Figure 7.79.)

When the projection of the major cutting edge into the back plane is not a straight line, then an array of flute spacings can be measured for many points of the cutting edge (e.g., to control the shape of the cutting edge of the drill shown in Figure 4.172). Figure 7.99 shows an example of measuring the flute spacing. Two heights, namely, $a_o{}_{i1}$ and $a_o{}_{i2}$, located on the major cutting edges on the same radius $r_i$ are measured, and then their difference is compared with the tolerance assigned on the cutting edge shape. Note that assessment of $a_o{}$'s when compared to the profile of the complicated shape cutting edges is the most proper way to compare the shapes of such cutting edges even when the most sophisticated drill measuring machines are used.

**Lip height variation**, $\Delta_{lh}$, is defined as article 05D in Table 7.37. Figure 7.100 provides a graphical representation of its measuring procedure for a two-flute drill. As can be seen, the projections AB and CD of the major cutting edges are brought into the same side of the axis of the datum by rotating AB about the axis of the datum by $180^\circ$. As such, projection AB occupies a new location A,B,
FIGURE 7.98  Showing an example of the definition of the major reference plane as the plane of radial symmetry at 180° for CoroDrill 860 drill. (Courtesy of Sandvik Coromant Co., Fair Lawn, NJ.)

FIGURE 7.99  Measuring the flute spacing for the points of the cutting edge located on the same radius $r_p$.

FIGURE 7.100  Measuring the relative lip height (the lip height variation).
The normal $\Delta_{lh}$ (the relative lip height) or axial $\Delta_{lh-ax}$ (the axial runout) can then be measured for any point of the projections A, B, and CD. The measured value can be compared to those assigned by the drill drawing and the referenced standards.

*Rake and clearance (primary) angles* are defined as articles 06D-11D in Table 7.37. The definitions of the planes of measurement and the corresponding articles are given in Chapter 4 (Section 4.5). It is discussed that the set of measuring plane (sometime called reference planes) should be defined for the considered point of the major cutting edge in T-hand-S. Figure 7.101 shows graphical representation of these planes for the point of consideration $i$. They are the normal, working, and back planes represented by sections A-A, B-B, and C-C, respectively (i.e., as should be shown in a tool drawing). These planes are related to the major reference plane as they are defined with respect to this plane as the following:

- The reference plane $P_{r,i}$ drawn through point $i$ is parallel to the major reference plane.
- The normal plane $P_{n,i}$ drawn through point $i$ is perpendicular to the major reference plane and to projection of the major cutting edge on which point $i$ is located into the reference plane. Note that if the cutting edge is not straight, then the tangent to the cutting edge at $i$ is considered.
- The working plane $P_{f,i}$ drawn through point $i$ parallel to the datum axis is perpendicular to the major reference plane.
- The back plane $P_{p,i}$ drawn through point $i$ perpendicular to the datum axis is perpendicular to the major reference plane.
- The cutting edge plane $P_{s,i}$ drawn through point $i$ is perpendicular to the major reference plane and contains the major cutting edge. Note that if the cutting edge is not straight, then the cutting edge plane that contains the tangent to the cutting edge at $i$ is considered instead of the actual cutting edge.

*FIGURE 7.101*  Geometrical characteristic of the major cutting edge at point $i$.  

![Diagram showing the geometrical characteristic of the major cutting edge at point $i$.](image-url)
It should be clear that not all the listed rake and clearance angles are indicated in a tool drawing. As discussed earlier, when the primary flank face is ground flat, then the normal clearance angle is indicated in the tool drawing. As such, this angle is the same (in T-hand-S) for any point of the major cutting edge. If needed, the angles in other reference planes can be calculated as discussed in Chapter 4 (Section 4.5). In the author’s opinion, advanced tool inspection machine having powerful on board computers should do such a calculation automatically and include the results (at least as an option) in the inspection report.

When the flank face is not flat, then the side clearance angle is indicated at point A, that is, at the drill corner as shown in Figure 4.55 in Chapter 4. In this case, the clearance angles are not the same for different points of the major cutting edge. Moreover, their distribution over this edge depends not only on the selected shape of the flank surface but also on the setting of the grinding fixture to generate the chosen shape of the flank face. The type of a particular surface selected to be a flank surface and setting parameters of the grinding setup (machine) are not indicated in the tool drawing. Therefore, it is important in this case to measure the clearance angle at least at three points of the major cutting edge, for example, at the periphery corner, at the middle, and at its inner end at the point of intersection of this edge with the chisel edge. In the author’s opinion, advanced tool inspection machine having powerful on board computers should provide distribution of the T-hand-S clearance angle over the major cutting edge. According to the author’s experience, when such information is available, the distribution of the clearance angle over the major cutting edge can be optimized to extend tool life and to prevent the interference of the flank face and the bottom of the hole being drilled in HP drilling.

The rake angle is not normally indicated in the tool drawing as discussed in Chapter 4 (Section 4.5.5) because of the following:

- It is assumed to be zero in T-hand-S for straight-flute drills.
- It is uniquely defined by the angle of helix for twist drills (see Equations 4.24 and 4.25).

For HP drills, however, the rake angle is indicated in tool drawings in the following cases:

- For plane rake faced twist drills (e.g., shown in Figure 4.59)
- For straight-flute and twist drills when the rake face is partially (the most common case) or fully modified by gashes (e.g., shown in Figure 4.51).

In the author’s opinion, advanced tool inspection machines having powerful on board computers should provide distribution of the rake angle over the major cutting edge as a part of the tool inspection report.

*Clearance angles of the secondary flank surface* are defined as articles 12D–14D in Table 7.37. They are measured in the same reference plane as those for the primary flank as shown in Figure 7.101.

The width of primary clearance, \( b_{fn} \), is defined as article 15D in Table 7.37. It is measured in the normal plane \( P_{n-i} \) as shown in Figure 7.101.

*Edge preparation* is defined as article 16D in Table 7.37. Figure 7.102 shows edge preparation (EP) types according to standard ANSI B212.4. The same types are defined by standard ISO 1832: 2004/2005. As discussed in Appendix C (Section C.4.2), out of these many standard types, more than 95% of drills and drilling tools receive a radius hone (Figure C.52). In real cutting tools, however, the theoretical profile does not turn into a smooth curve with a defined radius. The real profile of the transitional curve between the rake and the flank faces is similar to that shown in Figure 7.101 (Detail D). Although such a curve is no longer an edge (by the edge definition as to be a line), it is conditionally called the cutting edge. This real profile is approximated by a certain radius \( R_{ce} \) and is regarded as the EP radius.
As discussed in Appendix C (Section C.4.2), high-end EP measuring machines/microscopes do this approximation automatically (Figures C.62 and C.63) or even are capable of selecting the best curve rather than an arc to approximate the real profile. Moreover, the complete transition surface over the entire length of the cutting edge can be visualized digitally (Figure C.65). Unfortunately, no metrology for these advanced parameters is available yet, and thus, these parameters are not parts of the tool drawing. These examples are just another confirmation to the author’s point that the level of the available manufacturing and inspection equipment is far ahead of the drill design practice and metrology defined by the international and national standards in various developed countries.

In the author’s opinion, the following additional parameters should be included in the drawing of the advanced drills and drilling tools:

1. The radius of the cutting edge with the tolerance so that the real transition surface should be then approximated by the radius. The radii of this surface then can be measured over the entire length of the cutting edge (Figures C.62 and C.64) and compared with the tolerance assigned.

2. The surface roughness of the transition surface that should be better than those on the rake and flank faces as the surface roughness improvement (on the tool contact surfaces) is the leading idea of EP (see Appendix C). Figure C.49 shows a comparison of surface roughness before and after applying EP. Therefore, the inspection of surface roughness of the transition surface should be carried out to assure that the intended effect of EP is actually achieved.

The discussed measurements can be used as the prime parameter in capability studies of various available and emerging EP techniques (Figure C.55). Consistency/processing time/cost/improvement in drill performance sequence can be evaluated to select the optimal parameters of EP and its most efficient technique/technology.

The ability to approximate the real shape of the transition surface shape by the best-fit curve can be used in the future to apply real metal cutting theory to the optimization of the contact conditions.
7.16.4 Articles Related to the Chisel Edge and Its Region

Chisel edge angle, $\psi_{cl}$, is defined as article 17D in Table 7.37. Figure 7.103a shows its graphical representation.

Chisel edge length, $l_{cl}$, is defined as article 18D in Table 7.37. It is measured in the back plane. Figure 7.103a shows its graphical representation.

Chisel edge centrality (symmetry), $\Delta_{cc}$, is defined as article 19D in Table 7.37. Figure 7.103b shows its graphical representation.

Centrality of the web, $\Delta_w$, is defined as article 20D in Table 7.37. It is measured in the back plane. Figure 7.103c shows the graphical representation of the principle of the measurement of this article for a two-flute drill. Points C and B on the projection of the major cutting edges into the back plane are points of intersection of the chisel edge with the major cutting edges, and the line through center O perpendicular to the major reference plane is parallel to line EF. As such, the web centrality is calculated as

$$\Delta_w = |l_{w1} - l_{w2}|$$

(7.18)

Alternatively, the web centrality can be represented as

$$\Delta_w = \frac{|d_E - d_F|}{2}$$

(7.19)

where

- $d_E$ is the diameter of the circle with the center at point O and passing through point E
- $d_F$ is the diameter of the circle with the center at point O and passing through point F as shown in Figure 7.103c.

The obtained result can then be compared with the standard and recommended tolerances shown in Figure 7.77.

Web thickness, $d_{wb}$, is defined as article 21D in Table 7.37. It is measured in the back plane. In Figure 7.103c, it is the distance between points E and F, that is, the web thickness is calculated as

$$d_{wb} = l_{w1} + l_{w2}$$

(7.20)

**FIGURE 7.103** Characteristics of the chisel edge and the web in the back plane measured with respect to the major reference plane: (a) chisel edge angle, $\psi_{cl}$, and chisel edge length, $l_{cl}$, (b) chisel edge centrality (symmetry), $\Delta_{cc}$, and (c) centrality of the web, $\Delta_w$. 
Alternatively, the web thickness can be represented as

\[
d_{wa} = \frac{(d_E + d_F)}{2}
\]

(7.21)

Mismatch of the intersection lines of the secondary flank faces with the primary flank faces, \(m_f\), is defined as article 22D in Table 7.37. It is measured in the back plane as shown in Figure 7.104b. Note that individual mismatches \(m_{f1}\) and \(m_{f2}\) measured in the back plane between points E and F and the major reference plane with the back plane can also be measured to correct a grinding flaw (Figure 4.121).

Chisel wedge angle, \(\nu_{cl}\), is defined as article 24D in Table 7.37. It is measured in the chisel edge reference plane that is defined as the reference plane revolved about the axis of the datum by angle \(\psi_{cl}\) with respect to the major reference plane as shown in Figure 7.105b.

**FIGURE 7.104** Graphical representation of (a) the radius of the end of the chisel edge and (b) mismatch of the secondary flank faces.

**FIGURE 7.105** Chisel edge reference plane and the chisel wedge geometry in this plane: (a) chisel wedge angle is equal to 180°, (b) chisel wedge angle is less than 180°, and (c) curved chisel edge.
As discussed in Chapter 4 (Section 4.5.8.2), when the flank faces of the major cutting edges (lips) have the same T-hand-S clearance angles \( \alpha_n = \alpha_w \) and the width of the primary flank is equal or greater than \( 2\alpha_n \), then the chisel wedge angle is equal to \( 180^\circ \) as shown in Figure 7.105a. As such, when drill first touches the surface of the workpiece, it does so (at least, theoretically) by the whole length of the chisel wedge as this edge is perpendicular to the axis of rotation and feed motion (the axis of the datum in the geometry considerations). Such an engagement often presents problems causing poor shape of the machined hole at the entrance. As discussed in Chapter 4, Section 4.5.8.3, significant improvement in drill self-centering and in geometry of the chisel edge is achieved when the width of the primary flank is equal to \( \alpha_n \), while the secondary flank plane is applied with the normal clearance angle greater than that of the primary clearance angle. As such, the chisel wedge angle is less than \( 180^\circ \) as shown in Figure 7.105b, so when drill first touches the surface of the workpiece, it does so (at least, theoretically) by the central point connecting two portions of the chisel edge. For this and many other similar designs of drills with some self-centering ability, the chisel wedge angle, \( \nu_{cl} \), should be measured. Note that this angle is not a part of the tool drawing as its particular value is an outcome of many other parameters of the tool geometry (Section 4.5.8.3). However, this angle can and should be used to compare different drill geometries in terms of their effect on the drill’s self-centering ability. The smaller the chisel wedge angle, the higher the self-centering ability of the drill.

For drills with not planar flank faces, the chisel edge is not a straight line as shown in Figure 7.105c. As can be seen, such drills possess some self-centering ability that, to the first approximation, can be characterized by the radius of curvature of the chisel edge projection into the chisel edge reference plane.

**Chisel wedge symmetry, \( \Delta_{clw} \),** is defined as article 25D in Table 7.37. It is measured in the chisel edge reference plane as shown in Figure 7.105b.

**Chisel edge rake angle, \( \gamma_{cl-n} \),** is defined as article 26D in Table 7.37. It is measured in the chisel edge normal plane, \( P_{n-cl} \), which is defined as a plane perpendicular to the projection of the chisel edge into the chisel edge reference plane. Figure 7.106 shows graphical representation of these planes for point G of the chisel edge when the chisel wedge angle is equal to \( 180^\circ \) (Figure 7.106a) and when this angle is less than \( 180^\circ \) (Figure 7.106b). The rake angle of the chisel edge is defined as the angle between the chisel edge rake face and the chisel edge reference plane measured in the chisel edge normal plane. When the rake face of the chisel edge is sharply curved as, for example, when formed by pits (Figure 4.128), the tangent to the rake face of the chisel edge at point of consideration (point G in Figure 7.106) is drawn, and the chisel edge rake face is considered as the angle between this tangent and the chisel edge reference plane.

![Figure 7.106](image-url)  
**Figure 7.106** Chisel edge rake and clearance angle representation: (a) when the chisel wedge angle is equal to \( 180^\circ \) (Figure 7.104a) and (b) when the chisel wedge angle is less than \( 180^\circ \) (Figure 7.104b).
Note that, besides very special cases, the rake angle of the chisel edge varies along this edge so that it should be measured at various points of this edge to assess the distribution of the chisel edge rake angle over this edge. This distribution is an important design parameter for HP drills as discussed in Chapter 4 (Section 4.5.8). Moreover, for partially split chisel edges (e.g., as shown in Figure 4.128), the various points of the chisel edge may have considerable different rake angles.

**Chisel edge clearance angle**, $\alpha_{\text{cl-n}}$, is defined as article 27D in Table 7.37. It is measured in the chisel edge normal plane, $P_{n-cl}$, as the angle between the flank face of the chisel edge and the tool chisel edge plane, $P_{s-cl}$, as shown in Figure 7.106. The tool chisel edge plane $P_{s-cl}$ drawn through point G is perpendicular to the chisel edge reference plane and contains the chisel edge.

In some particular cases, the chisel edge is not straight so the cutting edge plane contains the tangent to the cutting edge at G that is considered instead of the actual chisel edge. If the flank face surface of the chisel edge is not planar, then the plane tangent to the curved rake face surface in the considered point on the cutting edge is used instead of the flank plane.

Note that, besides very special cases, the clearance angle of the chisel edge varies along this edge so that it should be measured at various points of this edge to assess the distribution of the chisel edge clearance angle over this edge.

**Length of the rake face of the split-point chisel edge**, $l_{sp}$ (only for generic design of split-point as one shown in Figure 7.107b), is defined as article 28D in Table 7.37. It is measured in the chisel edge normal plane as shown in Figure 7.107a. Note that $l_{sp}$ varies over the chisel edge as can be seen in Figure 7.107b, so its distribution over this edge should be measured (the details are discussed in Section 4.5.8).

**Transition radius between the rake face of the chisel edge and the chisel edge split gash**, $r_{sp}$, is defined as article 29D in Table 7.37. It is measured in the chisel edge normal plane as shown in Figure 7.107a.

### 7.7.16.5 Articles Related to Gashes

There are a number of ways to dimension gashes applied to the drill point to modify the rake faces of the major cutting edges (called web thinning) and/or to modify the extent and rake geometry of the chisel edge as discussed in Chapter 4. As a gash is normally a 3D article, its proper dimensioning and tolerancing present some problems in the tool drawing and thus in tool inspection.
One of the common ways to dimension gashes is by assigning the gash direction in the major reference plane by the gash reference angle (shown in Figure 4.51 as the web thin heel angle) and by the gash inclination angle in the back plane shown as the web thin notch face angle as shown in Chapter 4, Figure 4.51. The gash parameters are the actual gash opening angle (shown in Figure 4.51 as the web thin notch angle) and the actual radius (shown in Figure 4.51 as the web thin wheel radius). The extent of the gash is controlled by the resultant web thickness (shown in Figure 4.51). Such a representation of the gash parameters is easier for tool manufacturing but is not exactly strict for tool drawing where the representation of the plane perpendicular to the general gash direction to show the actual gash opening angle and radius is rather difficult.

Figure 7.108 shows one of the feasible ways of representation of gashes as, in the author’s opinion, the tool designer/developer should put in the drawing the parameters essential to the tool performance and the process development/manufacturing specialist should figure out how to translate these parameters into the position and shape of the grinding wheel. Once being a difficult job attainable only to few high-qualification specialists, this problem nowadays becomes almost routine as drills are designed using a 3D CAD program, so drill digital CAD model can be directly uploaded into a CNC grinding machine so that the grinding wheel shape is selected properly, its radius is dressed correspondingly, and it is positioned in the correct way with respect to the drill while applying the gash.

In Figure 7.108, the direction of the gash in the back plane (the face view of the drill) is defined by the gash face angle, $\psi_{gb-b}$, and the gash inclination angle, $\lambda_{gb-b}$, while its extent is determined by the distance $a_4$ from the CL to the beginning of the gash radius that, as discussed in Chapter 4, is an important drill design parameter. The radius of the gash in the back plane, $r_{gb}$, is also a part of the tool drawing. In the major reference plane, the gash plane angle, $\psi_{gb-r}$, is defined. If more than
one gash is applied (e.g., as shown in Figure 7.92), they should be dimensioned correspondingly. Therefore, these parameters are included in Table 7.37:

- **Gash face angle**, $\psi_{gb-b}$, is defined as article 30D in Table 7.37. It is measured in the back plane as shown in Figure 7.108.
- **Gash inclination angle**, $\lambda_{gb-b}$, is defined as article 31D in Table 7.37. It is measured in the back plane as shown in Figure 7.108.
- **Extent of the gash past the CL**, $a_4$, is defined as article 32D in Table 7.37. It is measured in the back plane as shown in Figure 7.108.
- **Radius of the gash**, $r_{gb}$, is defined as article 33D in Table 7.37. It is measured in the back plane as shown in Figure 7.108.
- **Gash plane angle**, $\psi_{gb-r}$, is defined as article 34D in Table 7.37. It is measured in the major reference plane as shown in Figure 7.108.

The listed articles should be measured for each gashed flute. Figure 7.108 shows a two-flute drill where two characteristics of these articles are to be measured and then compared with the tolerances assigned (hopefully) in the tool drawing.

It is common in the tool industry, however, to show other parameters of the gashes also shown for reference in Figure 7.108. The most common are the web diameter, $d_{wb}$, as determined by gashes and the beginning of the gashes on the tool rake face (distance $a_3$). The latter is often shown from the drill periphery corner (point A in Figure 7.108). If shown in the considered tool drawing, these parameters can also be included in Table 7.37. In the author’s opinion, being important, these parameters are outcomes of gashing. In other words, it is difficult to figure out the setting of the gash grinding wheel using these parameters.

### 7.7.16.6 Articles Related to the Coolant Holes

Two important articles related to the coolant holes location are measured in the back plane:

1. Location diameter, $d_{bd}$ (also known as the bolt diameter in the carbide rods, $D_{TK}$, as discussed in Section 6.3.3.10), is defined as article 35D in Table 7.37. It is measured in the back plane as shown in Figure 7.109.
2. Coolant holes location angle, $\psi_{ch}$, is defined as article 36D in Table 7.37. It is measured in the back plane as shown in Figure 7.109.

![Parameters defining the location of the coolant holes.](image)
Dealing with Nonincluded Articles

It should be clear that not all the articles and their definitions are included in Tables 7.36 and 7.37 as there are can be other design (dimensional and geometrical) articles and their characteristics assigned by the tool drawing for a particular drill. When dealing with such an article, the following sequence is recommended:

1. Define the datum feature(s).
2. Define the standard reference planes (e.g., the major reference plane, the back plane, the working plane) if the article is datum-related.
3. Define the article in the manner similar to definitions provided by Tables 7.36 and 7.37.
4. If needed, provide a simple sketch to facilitate developing its measuring procedure.

Dimensional Inspection (Metrological) System: Planning Stage

This stage has two prime objectives/outcomes:

1. Developing tables similar to Tables 7.36 and 7.37 where the non-datum and datum-related features particular to the drill to be inspected are listed with their proper definition. All the issues related to the listed articles and their characteristics should be resolved together with the drill design development and manufacturing/in-process inspection specialists.
2. Selecting the measuring equipment. Inspection and measuring equipment selected to carry out inspection include all of the tools and devices that are used to verify that a drill’s characteristics all conform to the tolerances required by the drill’s design/drawing. If needed, special inspection/measuring metrological procedures particular to the selected equipment should be developed.

The details of the first objective are discussed in the previous section where the definitions and graphical representations of the non-datum and datum-related features of the drill to be inspected are explained. Achieving the second objective requires some knowledge of drill gages in terms of their capabilities to achieve the required accuracy of the measurements.

Simple Gages

Various handheld gages including calipers and micrometers can be used to measure some geometrical parameters of drills. Such gages are widely available at small costs. Figure 7.110 shows an example of using a simple gage to measure some basic parameters of a drill. Figure 7.111 shows some drill gages available in the market. As can be seen, all these gages are hand gages. Their advantages are low cost and simplicity. Their disadvantage is accuracy of measurements. In the author’s opinion, this kind of gage is the best choice if one wants to attempt an ambitious home-improvement project. They are totally unacceptable for inspection of HP drills.

There are a number of simple optical devices for inspection of some features (articles) of the tool geometry. Among them, optical drill analyzers by Titan Co. (model OTWD-1 and OTWD-2) are most universal. The optical drill geometry analyzer shown in Figure 7.112 is an inexpensive optical instrument available for checking twist drill geometry after re-sharpening. Whether drill sharpening is done using a simple manual fixture or on a more expensive drill sharpening machine, it provides a quick and easy way of checking the results. The 5x magnification of the two basic optical units with their built-in scales helps one to insure that all angles are properly ground, web thickness is correct, and the point is properly centered. The use of the unit is simple as the drill to be measured is located into the basic V-block that is part of the unit. This centers it in relation to the built-in scales of the magnifiers. The unit is intended to measure the four components of the
twist drill geometry: the point angle, axial clearance angle at the drill corners, chisel angle, and web centrality. The accuracy of such measurements however is not nearly sufficient (approximately 20 times coarser than required) for HP drills. Being important parameters of the drill geometry for job-shop applications, the listed four parameters are only a small portion of the parameters listed in Tables 7.36 and 7.37.

7.7.17.2 Optical Microscopes/Measuring Systems

Inspection and measurement of HP drills go beyond handheld gages to include CNC drill inspection machines able to execute programmed measurement routines, laser micrometers (in- and post process) and contact and noncontact profilometers that measure surface roughness, and microscopes. The modern inspection systems include optical comparators, toolmaker microscopes, and digital microscopes designed especially for tool geometry inspection. Being different in the design, tool-holding, and measuring principles, these units allow to measure practically all non-datum articles listed in Table 7.36.

One of the major concerns in choosing which optical system is best for a particular shop is the cost. Optical including stereo microscopes are just for inspection, making them less expensive. They range from about $2,000 to $5,000. With a digital camera and some software, the cost increases to $12,000 or $15,000. Measuring microscopes start at about $15,000 and can go as high as $80,000, depending on options. Vision measurement systems start at about $40,000, but they can reach $200,000 or higher. Some pricing crossover exists between the measuring microscope and the vision measurement system categories. For example, depending on the shop’s needs, it might be worth looking into spending $40,000 or $50,000 on an automated vision system versus $60,000 on a manual measuring microscope. But with many different options available, the most important consideration is finding the right solution to suit the shop’s measurement needs assuring that the measuring system has the needed accuracy, range of drill diameters, and be suitable to inspect critical articles.

FIGURE 7.110 Measuring drill parameters with a simple gage: (a) measuring the length of the major cutting edges (lips), (b) measuring the angle of helix, and (c) measuring the chisel edge angle (length).
For years, the optical comparator was the most commonly used device for tool inspection. The comparator offered reasonably priced technology for checking drills by profile projection. Figure 7.113 shows an optical comparator as a detachable unit of a drill grinding machine. As can be seen, the location of the tool allows measurement in the major and chisel edge reference planes (e.g., shown in Figure 7.107). As such, many important parameters in these reference planes, for example, the point angle, lip height variation and length, and the point angle of the chisel edge, can be measured with reasonable accuracy while the drill is still in the grinding machine.

In modern designs of optical comparators as shown in Figure 7.114, the use of surface illumination such as LED illumination, digital readout and digital cameras, and edge finding further enhanced the measuring capabilities of comparators. Such units are built on a rigid cast base to assure metrological stability. Telecentric optics yield crystal-clear upright and reversed images as 10×–100× magnification lens. Motorized XY motion further enhances the accuracy of measurements. When one understands the definitions given in Tables 7.36 and 7.37, he or she can design and make the corresponding tool-holding fixture which enables to inspect of many articles listed in these tables with high accuracy.
FIGURE 7.112 Optical drill analyzer OTWD-2.

FIGURE 7.113 Optical comparator as a detachable unit of a drill grinding machine.

FIGURE 7.114 Modern computerized comparator with a digital camera.
Tool makers microscope (TMM) is another common device to inspect cutting tools. TMM is a type of a multifunctional device that is primarily used for measuring tools. These microscopes are widely used and commonly seen inside machine and tools manufacturing industries and factories. The main use of TMM is to measure the shape, size, angle, and the position of the small components that fall under the microscope’s measuring range. Modern TMMs are outfitted with a CCD camera that has the ability to capture, collect, and store images into specialized computer software.

The glass grading and optics system make TMMs fully functional because, whatever is being viewed under these microscope parts or precision instruments, it is important that the objectives and the eyepiece lenses are made of fine-quality glasses only. These essential parts are what makes the device very durable and gives it the ability to withstand the wear and tear associated with the everyday stress of factory usage.

Some TMMs are equipped with a cross hair reticle on the eyepiece, coupled with a protractor on the tube. High-end TMMs use semiconductor laser devices as directors. Instead of the cross hairs, a red point is virtually marked on the microscope’s working surface in order to locate the parts that have to be measured by the microscope. The CCD imaging system can also be used as a measurement system as well. This is another advanced feature of the newer versions of toolmaker’s microscope models. A CCD camera that has the ability to measure diameters and distances provides additional conveniences in measurements. The magnification power of TMMs is nothing better than a regular compound microscope. In fact, it has a total magnification power up to 80×. This is due to the fact that these microscopes require good working distances of around 100 mm.

Figure 7.115 shows Mitutoyo 176-808A TMM with digimatic micrometer heads, 30× magnification, although some other TMMs are widely used. Commonly, TMM consists of the cast base, the main lighting unit, the upright with carrying arm, and the sighting microscope. The rigid base is resting on three-foot screws by means of which the equipment can be leveled with reference to the built-in box level. The base carries the coordinate measuring table and consists of two measuring slides: one each for X and Y directions and rotary circular table provided with the glass plate.

**FIGURE 7.115** Mitutoyo 176-808A TMM with digimatic micrometer heads, 30× magnification.
The slides are running on precision balls in hardened guide ways warranting reliable travel. Two micrometer screws with normally digital readouts assure measuring range of 0–25 mm and permit the measuring table to be displaced in the X and Y directions.

TMM is suitable for the following fields of applications:

- Length measurements in Cartesian and polar coordinates
- Angle measurement of tools, threading tools, punches and gages, templates, etc.
- Thread measurements, that is, profile, major and minor diameters, thread angle, lead, and profile position with respect to the thread axis
- Simple measurements of the parameters of cutting tool geometry
- Tool wear analysis and measurements, including wear pattern parameters.

Stereo microscopes provide a 3D view of the part (a cutting tool) and generally are used for visual inspection, not measurement. A stereo microscope works like the human eye, which means it has good depth of field, that is, one can see a 3D image magnified maybe 20× or 30× on a stereo microscope. It is most common to look through the eyepiece when using stereo microscopes, but they also are available with a digital camera to view the part on a computer screen. Shops that want to document a defect can do so by adding the camera and software.

Stereo microscopes are appropriate for the shop floor as well as the QC lab to inspect a tool after it has been machined. It is popular for users to look for strange artifacts, defects or burrs, and the surface condition of the tool. Another application for stereo microscopes is cutting tool wear inspection. It is used in wear studies including studies of coating suitability, machining regime optimization, and influence of the drill geometry parameters on wear.

While rough measurements can be performed on stereo microscopes, measuring microscopes, or toolmakers’ microscopes, produce a 2D image for more critical measurements. While an eyepiece is typically used with stereo microscopes, with measuring microscopes, the image is usually shown on a computer screen using a video camera. The measurement is made in the software using the image from the camera, not using the part itself. It is easier for tool inspectors because they do not have to strain their eyes as they can move the microscope stage in the X and Y directions and see the tool moving around while looking comfortably at the computer screen. The inspector takes a series of data points, say around a circle, and the software takes those points and calculates the circle that those points make up. He or she can take a number of points and measure diameters, lengths, arcs, and angles. If documented inspection is required, the data can be output in a spreadsheet or SPC program.

### 7.7.17.3 Specialized Measuring Microscopes

The need for a special microscope to control essential geometrical parameters of the drill was always there. Figure 7.116 shows the pure mechanical measuring fixture according to US Patent No. 3,414,979 (1968). In this inspection fixture, the drill 1 is located on the precision rollers 2 so that the proper datum location of the drill is assured. Dial 3 is to indicate the drill point angle, while adjusted dial indicator 4 is to measure the lip height variation. Dial indicator 5 is connected through a movable carriage 6 (Section A-A) to measure drill runout. The dial indicator 7 is to measure back taper (Section B-B) and other parameters of the drill margins. It is clear that the use of such a fixture requires clear understanding of the features (articles) to be measured/inspected and high skill of the operator for its proper use. It can be considered as a beta-version of modern special microscopes. Note, however, that not all modern microscopes are meant for and thus capable of measuring features that the fixture shown in Figure 7.116 can, for example, runout.

In the author’s opinion, the HP drill manufacturers and users should have a measuring microscope for proper control of HP drill quality, non-datum article measurements (see Table 7.36), wear studies/control, etc. A few models of such microscopes developed especially for inspection of axial tools are available in the market. Figure 7.117 shows Zoller promBasic measuring microscope as an example. Although the appearance of such microscopes from various manufacturers as EuroTech
Drills: Science and Technology of Advanced Operations

(e.g., PG1000), Zoller (see Figure 7.117), Walter, etc., looks almost the same for a casual observer as it includes the swivel base with a V-block, LED lights, camera with the objective, $X$ and $Y$ (horizontal and vertical) precision motions, and monitor with mouse-click on-screen measurements, their characteristics and measuring abilities are not nearly the same. The only common feature of the microscopes available today is that none of them has a manual containing instructions on what features of a drill


FIGURE 7.117 Zoller promBasic measuring microscope.
Metrology of Drilling Operations and Drills can be measured and corresponding step-by-step procedures. As a result, some end users developed a number of measuring procedures according to their perceptions and understanding of what and how to measure that may or may not be in lines of the definitions provided by Tables 7.36 and 7.37. The rest, however, just use this piece of equipment for visual observation of axial tools in manufacturing and for wear analysis. Therefore, a need is felt to provide explanations on the essential features of such equipment and practical recommendations for their proper use.

Figure 7.118 shows the proper architecture and basic positions of the tool measuring microscope to facilitate the discussion on the procedure of measuring the common parameters (articles) of the drill geometry discussed earlier in this chapter. Each of the positions shown in this figure corresponds to a certain plane of measurement, and thus the parameters (articles) in this particular plane can be properly observed and measured. Note that the procedure discussed in the following text is universal, that is, applicable to any tool measuring microscope available today provided that the swiveling base table of the microscope can be brought to all positions shown in Figure 7.118.

As shown in Figure 7.118, four (for each cutting edge) basic positions of the swiveling base are selected. Position 1 represents the measurements in the back plane, position 2 represents the reference and working planes, position 3 represents the cutting edge plane, and position 4 represents the normal section plane. Corresponding non-datum articles listed and defined in Table 7.36 can be measured in these planes.

The inspection procedure begins with setting the tool in position 1, so it is considered as the starting position. The tool in the V-block and objective are brought together to an appropriate scale where the tool should be at magnification allowed by the monitor size. The tool should be cleaned from small debris using edge cleaning putty. Figure 7.119a shows the view that one should obtain in this position. First, the tool is visually examined for obvious flaws and chipping, excessive asymmetry of features (e.g., the position of the coolant hole with respect to the major cutting edges [lips]), coarse surface finish of the flank faces, etc. Second, the scale is applied (Figure 7.119b) and the following articles listed in Table 7.36 can be inspected: 08N, margin width; 09N, body clearance diameter; and...
14N, coolant hole diameter. The following articles listed in Table 7.37 can be inspected: 17D, chisel angle $\psi_{cl}$ (approx.); 18D, chisel edge length $l_{cl}$; 21D, web thickness, $d_{wb}$; 22D, radius of the end of the chisel edge, $r_{cl}$; 23D, mismatch of the intersection lines of the secondary flank faces with the primary flank faces, $m_i$; 30D, gash face angle, $\psi_{gh.b}$; 31D, gash inclination angle, $\lambda_{gh.b}$; 32D, extent of the gash past the CL, $a_4$; 33D, radius of the gash, $r_{gh}$; 34D, bolt diameter, $d_{bd}$; and 35D, coolant holes location angle, $\psi_{ch}$ (approx.). Note that the flank and chisel edge wear (see Section 2.6.4) is not measured in the back plane although practically all literature sources depict such wear as viewed in the back plane.

The next step in tool inspection is to bring the swiveling table in position 2a. Normally, the angular scale around the swiveling table is provided as shown in Figure 7.120. This enables the user to set the current position of the table on the scale to any chosen angle with approx. 1° accuracy. Figure 7.121a shows the view that one should obtain in this position. First, the tool is visually examined for obvious flaws in the same way as in position 1. Particularly, surface roughness on the drill margin and chipping of the side cutting edge are assessed.
The following characteristics of the articles can be measured: (1) axial clearance angle at drill corner in the manner discussed in Section 7.7.12 and shown schematically in Figure 7.87 (Figure 7.121b shows actual measurement of this article) and (2) axial rake angle at drill corner. For a straight-flute drill it should be zero if no special measures to modify the rake face geometry, is taken. For a twist drill, this angle should be equal to helix angle, \( \omega_d \), (article 25N—in Table 7.36).

The characteristics of other articles listed in Tables 7.36 and 7.37 can be measured: 03N, flute length; 04N, active flute length (approx.); and 05N, length of the body relief. If a drill has steps (stages) with secondary and other cutting edges, then the step distances can be approximately measured. Other useful parameters of articles not listed in Tables 7.36 and 7.37 can also be measured. For example, Figure 7.122 shows re-sharpening notches in front of the second drill step.

**FIGURE 7.121** View in position 2a: (a) view for general assessment and (b) axial clearance angle at drill corner measurement.

**FIGURE 7.122** Showing the re-sharpening notches in front of the second drill step.
Counting these notches, one can conclude that the drill was re-sharpened eight times. If the parameters of the body undercut in front of second, third, etc., cutting edges are important, they can be assessed by increasing the zoom.

The discussed measurement/assessments are related to the right major cutting edge. Rotating the swiveling table by 180°, that is, to position 2h, the same measurement/assessments can be made for the left major cutting edge. When a drill has more than two teeth (the flute), all measurements/assessments were carried out in position 2a by rotating drill by the central angle between teeth (the flute).

Then the drill is rotated by 90° (for two-flute tool) or by 180/\(z\) angle (where \(z\) is the number of cutting teeth [flutes]) clockwise about the axis of the datum (the axis of the shank). Figure 7.123a shows the view that one should obtain in this position. First, the tool is visually examined for obvious flaws in the same way as in position 1. The following parameters of the corresponding articles in the major reference plane can be measured (see Tables 7.36 and 7.37): 01N, drill diameter (approx.); 02D, point angle \(\Phi_p\); 24D, chisel wedge angle, \(\nu_{cl}\) (approx.); and 34D, gash plane angle, \(\psi_{gh-r}\). Figure 7.123b shows an example of measuring the point. If the drill is made with generic design of split-point drills it is further rotated clockwise by chisel angle \(\psi_{cl}\) to measure the length of the rake face, \(l_{sp}\) (article 28D in Table 7.38).

The next step in tool inspection is to bring the swiveling table to position 3a. The angle of the swiveling table is set to be a half-point angle \(\phi\) obtained by dividing the point angle measured at position 2 by two. The achieved drill position with respect to the camera objective corresponds to the cutting edge plane. Figure 7.124a shows the view that one should obtain in this position. First, the tool is visually examined for obvious flaws in the same way as in position 1. Particularly, surface roughness of the drill prime flank surface and microchipping of the cutting edge are at various magnifications. If edge preparation (see Section C.4.2) is applied and when the optical zoom of the microscope allows, the conditions and uniformity of the applied edge preparation can also be assessed.

Drill flank wear is assessed/measured in this (the cutting edge) plane so that this plane is the most important in any assessment of drill performance. If optical zoom of the measuring microscope allows, a detailed analysis of wear topography can be attempted to determine the root cause

![Figure 7.123](image-url)
of flank wear (Chapter 2). Figure 2.51 in Chapter 2 shows various topographies of the drill flank wear in the cutting edge plane. Figure 7.124b shows an example of the measuring of the drill corner wear. Note that HSS and sintered carbide drill wear is much more great than that of PCD drills. So if the wear/damage of PCD tool is to be attempted, much greater optical zoom is required on the measuring microscope.

If a drill has step(s), then the second (third, fourth, etc.) cutting edge and the discussed parameters in the cutting edge plane can be assessed/measured. If a step has the point angle equal to that of the first stage, then the carriage is moved horizontally and zoomed on the cutting edge of the second or other cutting edges. If a stage has the point angle different than the first stage, it is measured in position 2, and the angular position of the swiveling table is set correspondingly by the half-point angle of the considered stage. Figure 7.125 shows the view in position 3 for the cutting edge of the second stage.

The discussed measurement/assessments are related to the right major and second cutting edges. Rotating the swiveling table by 180°, that is, to position 3b, the same measurement/assessments can
be made for the left major and secondary cutting edges. When a drill has more than two teeth flutes, all measurements/assessments are carried out in position 3a by rotating drill by the central angle between teeth flutes. If a drill has step(s), then the second (third, fourth, etc.) cutting edge and the discussed parameters in the cutting edge plane can be assessed/measured.

The next step in tool inspection is to bring the swiveling table to position 4a. The angle of the swiveling table is set to be a half-point angle $\phi$ with respect to position 2. Figure 7.126a shows the view that one should obtain in this position at high zoom. Position 4a corresponds to the normal plane for the right major cutting edge (see Figure 7.101) so that the normal clearance angle of the major cutting edge can be measured in the manner shown in Figure 7.126b. This angle is important as it is normally indicated in tool drawings.

The discussed measurement of the normal clearance angle is related to the right major cutting edge. Rotating the swiveling table, to position 4b, the normal clearance angle can be measured for the left major cutting edge. When a drill has more than two teeth flutes, all measurements/assessments were carried out in position 4a by rotating drill by the central angle between teeth flutes. If a drill has step(s), then the second (third, etc.) cutting edge and the discussed parameters in the cutting edge plane can be assessed/measured. If a step has the point angle equal to that of the first stage, then the carriage is moved horizontally and zoomed on the cutting edge of the second or other cutting edges. If a stage has the point angle different than the first stage, it is measured in position 2, and the angular position of the swiveling table is set correspondingly by the half-point angle of the considered stage.

The discussed procedure is related only to some standard articles of a drill. The parameters of other articles that may be particular to a given drill design can be measured in the similar manner knowing the corresponding plane of measurements and adjusting the swiveling table and drill angular position correspondingly. Before measuring/assessing any drill parameters, this parameter and the plane of its measurement should be clearly defined by the tool drawing because one can only inspect parameters indicated in this drawing.

The discussed procedure implies that a measuring microscope should (1) have sufficient space to bring a drill in the positions shown in Figure 7.118 (unfortunately, this is not the case in some modern
measuring microscopes where the angle of the swiveling table rotation is restricted by ±90°, that is, it is impossible to bring a drill to position 4. The author advises not to use such microscopes as they severely restrict user’s ability to measure parameters of the essential articles listed in Tables 7.36 and 7.37) and (2) have a great optical zoom. This is particularly important for HP drills including PCD drills. While the former is rather obvious, the latter is not well understood so it requires further clarifications.

While TMMs normally have 30× lens (15× eyepiece lens and 2× objective lens) zoom, the measuring microscopes have 6x–15x optical (called—lens) zoom. The manufacturers of the measuring microscopes claim that they have a great digital zoom so that the total magnification should be sufficient for even tiny drill inspection. This common misconception lasts until a blurry image of the drill appears on the monitor no matter how many megapixels the digital camera has.

What is the difference between optical zoom and digital zoom? The short answer is optical zoom = resolution and digital zoom = cropping. Optical zoom is lens-assisted magnification. It is usually measured as the ratio of maximum focal length possible to minimum focal length possible. For example, a 28–110 mm lens will have an optical zoom of 110/28 = 4×. A higher optical zoom is a better choice because a lens can have distortions toward the low or high focal lengths.

Digital zoom is just cropping the image and zooming in (digitally or pixel wise) to the interesting area. It is more like a digital technique and misinforms the customer. This is because digital zoom is not really zoom, in the strictest definition of the term. What digital zoom does is enlarge a portion of the image, thus simulating optical zoom. In other words, the camera crops a portion of the image and then enlarges it back to size. In so doing, image quality deteriorates.

What, therefore, is the rule of thumb, when it comes to using zoom? Here it is: always use optical zoom. Whatever you do, buying a microscope or digital camera, choose one that warns you that you are about to use digital zoom or that allows you to disable digital zoom (most do). There is no point in comparing digital zoom with digital zoom or optical zoom with total zoom. Always compare optical zoom with optical zoom.

While considering implementing a measuring microscope, end users should clearly understand the following:

1. Only some of the datum-related articles listed in Table 7.37 can be measured. This is because although the tool shank as the common datum is located in a V-block and tightly pressed against it so that the tool location is correct, the axis of the shank is not visible on the screen, so no measurement of the tool can be taken with respect to this datum.
2. The image on the screen is not digitally correlated with the on-screen measurements. In other words, no edge finder is available for precision measurements as the user using the mouse defines the lines and then measures the distance and angles between these lines. While for many angular measurements such precision is acceptable, linear measurements accuracy may not be sufficient for some non-datum features.

7.7.17.4 CNC Measuring Systems

To evaluate the existing CNC measuring/inspection systems, one needs to understand what would be the ideal measuring/inspection system. The following wish list describes the desirable features of such a system:

1. It should be capable to measure/inspect the articles described in Tables 7.36 and 7.37 plus parameters of drill geometry in T-mach-S and T-use-S described in Chapter 4.
2. It should be suitable for fully automated measurements removing the subjectivity of manual and semi-manual measurements. Users are just teaching it basically to go around the drill in relation to the CAD drawing and tell the machine what to measure. The program is just recording the steps one is making in measuring various articles according to the drill drawing. Then this program can be saved and can be run over and over. The program also can be easily edited to make other programs from it.
3. To facilitate the measurements/inspection, the system should include a library of common and standard features (articles) and definition/measuring functions. Corresponding tool geometry standards and nonstandardized parameters should be clearly indicated to help a potential user to make the proper choice of a particular built-in procedure.

4. It should include a combination of very low-distortion optics, precision spindles, rigid frames, and high-resolution video that can be processed by the software to make automatic measurements.

Technically, the ideal system should be able to carry out the following measurement sequence:

1. **Hang** the drill in midair.
2. Scan its fully digital image.
3. Determine the datum according to the drill drawing. For drilling tool, the common datum is the axis of the shank.
4. Measure the datum geometry and store the information. The shank cylindricity and surface roughness are measured.
5. **Idealize** the datum, that is, to make it perfect digitally. The actual shank axis is determined and postulated that this axis is ideal, that is, having no deviation.
6. Digitally correct the scanned image accounting on the corrections made to the datum. As a result, a completely new digital image of the drilling tool is obtained.

The ideal system should allow to define the purpose of measurement. Normally, there should be two options: (a) inspection and (b) research. The sequence of measurements depends upon the chosen option.

The inspection option should allow the following:

1. Load the CAD drawing into the measuring system computer. Note that this drawing should contain well-defined articles and their parameters with tolerances to be inspected. Moreover, they should be clearly specified according to the corresponding tool geometry and tool drawing standards. According to the author’s experience, this is the major problem as the articles and their characteristics in many tool drawings are defined using the *free-lens* style, that is, according to the designer’s understanding and interpretation of drill features.
2. The measuring system determines articles to be measured from the loaded tool drawing and carries out the corresponding measurement using the corrected digital image of the tool.
3. The inspection protocol is created where the measured and drawing parameters are listed and compared to determine which of the measured parameters are is and which are out of the corresponding tolerance zones assigned by the tool drawing tolerance zone. Such a protocol lists the parameters defined by the drawing versus measured parameters showing the location of the measured parameters with respect to the tolerance zone. For convenience, the out-of-tolerance measured parameters should be highlighted, for example, indicated by a different eye-catching color.

In the research option, the modified digital image of the tool is dealt with. Depending upon the purpose of a particular research, some measurements defined by the tool geometry standards and thus built-in in the measuring system can be attempted as well as the measuring of some parameters in nonstandard reference plane. The system should be able to help the user to determine any section or reference plane easily with respect to the datum. As such, the determination of the geometry parameters in T-mach-S and T-use-S (see Chapter 4) should not present any problem, which helps to optimize the parameters in T-hand-S, that is, those shown in the tool drawing.

Another important aspect of such a system is to measure 3D wear topography and volume of the work tool material through comparing digital images before and after tool run. As discussed by the author
earlier (Astakhov 2004), such an assessment is the most objective way to evaluate various tool materials and optimality of the tool design. Collecting and storing the data can also help to evaluate any change in the drilling operation should unexpected tool failure(s) occur due to, for example, problems with castings or the machine spindle. Besides, the machinability of the work materials can be compared objectively.

Moreover, the ideal CNC measuring system should be able to simulate drilling, that is, it should be able to rotate and feed the drill into a digital workpiece. The user should be able to make any desirable cross section of the digitally formed assembly. The following objectives of such simulations are of prime importance in the drill design and optimization:

1. Verification of the absence of interference of the drill flanks with the bottom of the hole being drilled in the manner shown in Figure 5.108.
2. Measuring arms of the unbalanced moments, for example, $a_{xub}$ and $a_{yub}$ shown in Figure 4.18; $a_{xub}$ and $a_{yub}$ shown in Figure 5.50; and $z_{a12}$, $z_{a23}$, $z_{b12}$, and $z_{b23}$ shown in Figure 5.101. These are needed for the detailed optimization of the tool geometry as discussed in Chapters 4 and 5.
3. Visualization and quantification of the topology of the bottom clearance space (see Chapters 4 through 6) for optimization of the flank surface design and geometry as well as for improving the MWF flows in this bottom clearance space (see Section 6.3.4).

Unfortunately, among CNC drill measuring systems available today, no one is capable of fulfilling the previously discussed wish list. Therefore, the suitability of the available and emerging systems can be assessed using the previously discussed requirements, that is, by the relative importance, measurement accuracy, measuring complexity, compliance with the drill geometry standards, and the extent to which a particular system complies with the listed requirements.

As not many systems of this caliber are available in the marketplace, such a ranking does not present a problem. What one needs is just to take Tables 7.36 and 7.37 and go over the articles listed in these tables with a sales/technical representative of a perspective measuring machine. In the author’s experience, only very few machines can survive such an assessment particularly when the datum-related articles are considered. There are a number of reasons for that. The most common are as follows:

1. No one CNC tool measuring machine is supplied with a manual that includes the definitions, drawing, principles, and examples of measurement of drill standard articles listed in Tables 7.36 and 7.37. A short training provided by the machine manufacturers is oriented mainly on the basic functions of the machine. As a result, many measuring machines in the tool manufacturers’ and users’ possession are used only up to 30% of their capability. Most of them are used as an expensive tool presetting machine as the inspection reports include only the diameters, runouts, and step lengths as discussed in Chapter 2 with examples.
2. Practically no one tool drill geometry measuring machine uses the standard terminology and thus definitions of drill articles listed in Tables 7.36 and 7.37. Rather, a number of own homemade terms are used with no proper definitions.

There are two basic CNC tool measuring machines available today in the tool industry, namely, the Zoller Genius 3 Pilot 3 (Figure 7.127a) and a family of the Walter Helicheck machine (Figure 7.127b shows Helicheck Plus L). The capabilities and accuracy of these machines are similar so that an inexperienced user may have a hard time to select which one of these two machines is the best choice for his/her measuring needs. The absence of manuals where the well-defined features of the tool geometry and dialed step-by-step procedures of their measurement using these machines just adds more challenge to the selection challenge. The ignorance of the basic articles and their standard definitions listed in Tables 7.36 and 7.37 from both sides of the fence does not help either.

The author’s analysis of the available CNC measuring machines shows that these machines are capable in principle (hardware-wise) of measuring practically all articles listed in Tables 7.36 and 7.37 as well as the articles in T-mach-S and T-use-S. The problem is that just few of them are set as the
standard functions, while most require additional programming. Such programming can be attempted using the graphics and definitions presented in this chapter with close collaboration with the specialists of the corresponding companies. The author considers such a situation as normal as CNC measuring machines are in the infant stage of their development and implementation in the cutting tool industry.

The great advantage of CNC measuring machines is proper definition and nullification of the datum. As discussed previously, the datum for any HP drill is the axis of its shank. This datum is considered as to be perfect so that other datum-related parameters of the tool design and geometry are measured with respect to this datum. In reality, however, nothing is perfect because the shank is mounted in the tool holder, which, in turn, is mounted on the measuring machine spindle. As a result, the datum axis of the tool may not be coincident with the axis of spindle rotation. As the measurement of the tool parameters is made with respect to the axis of rotation, the measured results cannot be compared with those assigned by the drawing with respect to the datum.

To resolve the issue, the datum nullification is the first mandatory step in the use of any CNC measuring machine. Figure 7.128 shows its realization on the Zoller Genius 3 Pilot 3 machine. As can be seen, the runout of the shank is actually measured. Then the result of this measurement is used for the corresponding correction of the measurement of other parameters of the drilling tool. Obviously, a drilling tool should be mounted in the tool holder so that a part of its shank can be measured, so the datum can then be nullified. Note that this step is not called properly as datum nullification, on the machine. Rather the term wobbling compensation is used.

Figure 7.129 shows the two basic locations of the measuring camera. These locations correspond to the positions 2ab and 1 shown in Figure 7.118, correspondingly. Therefore, the non-datum and datum articles can be inspected/measured in the tool reference, back and working planes to inspect practically all features listed in Tables 7.36 and 7.37. As mentioned previously, some of these are built-in features of the machine, while others required additional programming.

Figure 7.130 shows an example of the menu where some of the built-in standard articles are listed. An article or series of articles can be selected by the machine operator so that the machine
FIGURE 7.128  Datum nullification position.

FIGURE 7.129  Basic locations of the measuring cameras with respect to the drill to be measured: (a) side position and (b) top position.

FIGURE 7.130  Example of the menu where some of the built-in standard features are listed.
can measure the features of the selected articles almost automatically. For example, if one selects to measure the normal T-hand-S clearance angle (wrongly termed as the axial relief angle on the menu), then the next menu for this article with some additional choices and visualization of the measuring flank face(s) appears on the monitor screen as shown in Figure 7.131. Although not obvious, the normal or axial clearance angle in T-hand-S for a selected point of the major cutting edge (lip) can be measured. For example, if one selects option *in relation to cone angle* (which actually stands for the drill point angle) as shown in Figure 7.131 then the normal clearance angle will be measured. Otherwise, the side (axial) clearance angle will be measured if this option is not selected.

Figure 7.132 shows the result of measuring the normal clearance angle at the selected point of the cutting edge. As can be seen, it is 7.8°. The machine, however, offers some additional useful options.

**FIGURE 7.131** Menu for measuring the normal clearance angle over the major cutting edge.

**FIGURE 7.132** Showing the result of measuring the normal clearance angle in the selected point of the cutting edge.
Two of these options are of prime interest. First, the digital image of the primary flank plane with 3D visualization of the clearance angle at the considered point of the cutting edge can be viewed and orbited by the computer mouse as shown in Figure 7.133. As such, one can see the variation (if any) of the normal clearance angle along the major cutting edge. Moreover, the ground texture of the primary flank face can be viewed at high magnification to analyze, for example, grinding marks and micro-defects. Second, the actual shape of the primary flank face in its section by the normal or orthogonal plane through the selected point of the major cutting edge can be viewed as shown in Figure 7.134.

As can be seen, the polynomial of the first order (a straight line) is selected to overlay the actual shape of the section of the primary flank. If the primary flank is not a plane but a 3D surface generated
by a CAM grind (e.g., a part of conical and helical surface as discussed in Chapter 4), then the line of its intersection with the normal or orthogonal plane through the selected point of the major cutting edge is a complicated curve. By selecting a suitable polynomial degree on the screen menu, one can obtain the best fit (approximation) of the flank face so that the normal to the flank face can be drawn and the proper clearance angle in the manner shown in Figure 7.91b can be measured. Such an operation, however, requires additional programming as it is not readily shown in the on-screen menu.

Unfortunately, a great number of articles listed in Tables 7.36 and 7.37 and crucial to HP drill performance are not built-in features that can be measured easily using a series of on-screen menus and options. An example is an attempt to measure the chisel edge centrality (symmetry) \( \Delta_{cc} \) as listed as article 19D in Table 7.37 and visualized in Figure 7.82. Figure 7.135 shows the screen and menu for the inspection of straight line symmetry that actually stands for flute spacing measurement as defined by article 4D in Table 7.37 and shown in Figure 7.97a. This is selected because it properly visualizes the web centrality. Figure 7.136 shows what should be considered in the centrality of the web measurement (unfortunately not actually highlighted on the monitor in the manner shown in this Figure). The chisel

![Screen and menu for inspection of straight line symmetry](image1)

**FIGURE 7.135**  Showing the screen and menu for the inspection of straight line symmetry used to visualize the web centrality.

![Web centrality measurement](image2)

**FIGURE 7.136**  Showing what should be considered in the centrality of the web measurement.
edge centrality (symmetry) $\Delta_{cc}$ is the distance between the datum and the actual chisel edge, that is, as perpendicular from the point designated as the datum in Figure 7.136 to the highlighted chisel edge. If the chisel edge would be highlighted and digitized in the manner shown in Figure 7.136, then the following articles listed in Table 7.37 can be measured in this drill position/screen menu: the chisel edge angle, $\psi_{cl}$ (17D); chisel edge length, $l_{cl}$ (18D); centrality of the web, $\Delta_{w}$ (19D); and web thickness, $d_{wb}$ (21D). The listed parameters are visualized in Figure 7.103.

### 7.7.17.5 Summary

Inspection and measurement of HP drills go beyond handheld gages to include CNC drill inspection machines able to execute programmed measurement routines. For years, the optical comparator was the most commonly used device for tool inspection. The comparator offered reasonably priced technology for checking drills by profile projection. The use of surface illumination, digital readout, and edge finding further enhanced the device.

However, the comparator lacked the ability of the naked eye or digital camera to see fine details in the geometry of cutting tools. The toolmakers’ microscope, with its linear scales and digital readout, was an improvement over the comparator, because tool images could be clearly seen and measured. But like the comparator, the toolmakers’ microscope was still a mechanical system limited to hardware upgrades. Like a comparator or microscope, a digital system uses ground glass lenses to magnify features on a tool. However, this type of system also uses a camera to convert the tool image to a digital image.

Early versions of monitor-based video inspection systems used black-and-white cameras, television monitors, and line generators. Measurements were made by linear scales and digital readout or by a pixel count from the monitor screen. In a video system, the number of pixels, or single dots on the monitor screen, determines the display resolution. With pixel count, cutting tools can be easily measured. And because tool images are in a digital format, they can be printed, but they lack supporting documentation such as $x$-axis, $y$-axis, and rotational measurements. In the past 10 years, the lower cost and higher sophistication of computers, video imaging cards (frame grabbers), and cameras have boosted the popularity and accuracy of CNC video inspection systems. System manufacturers can use frame grabbers to display live tool images on a computer monitor screen and write custom programs that increase the speed and accuracy of video inspection. The greatest advantage of CNC measuring machines is proper definition and nullification of the datum that sets such machines apart from any other HP drill measuring devices. The introduction of this feature opens a new era in HP drill inspection/measurement that, for the first time in the history of drill measurement, can be carried out exactly according to the proper tool drawing. Modern advanced CNC measuring machines can, in principle, inspect/measure all non-datum and datum articles required for HP drills. Although it is true, three issues remain that slowdown implementation of CNC measuring machines and thus HP drills:

1. *Proper HP drill drawing*. This issue is discussed in Chapters 4 and 5. The articles and their parameters listed in Tables 7.36 and 7.37 should be properly and clearly indicated in drill drawings. Otherwise no one knows what to inspect/measure. The author pointed out earlier (Astakhov 2010a) the basic requirement to tool drawings and presented a number of examples. This chapter includes examples of the proper representation of the datum and non-datum features listed in Tables 7.36 and 7.37 in HP drill drawings.

2. *Outdated standards* (ISO, DIN, etc.) for parameters of the articles listed in Tables 7.36 and 7.37. This chapter provides basic recommendations for the proper selection of such parameters for HP drills.

3. *The limited library of common features/articles to be measured almost automatically*. Using CNC measuring machines. In the author’s opinion, it is very difficult to justify a $200,000 nonproduction machine that then is used only to 30% of its capacity and has limited built-in standard articles to be measured automatically, suffers from nonstandard/improper terminology, and has no useful user-friendly manual.
In the author’s opinion, the listed issues can be resolved in relatively short time so HP drilling tools can be properly inspected. It should provide a significant boost for wider implementation of HP drills.

REFERENCES


