Chapter 9
Processes with Rotational Primary Movement

Machining methods with geometrically defined cutting edges in which the main movement is rotational are subdivided in accordance with Fig. 9.1 into

- turning,
- milling,
- drilling and
- sawing.

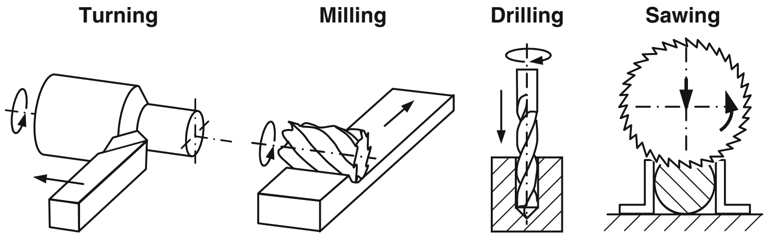


Fig. 9.1 Classification of processes with rotary primary movements

A propos of sawing, it should be noted that a purely rotational main movement is only executed in the case of circular sawing. In the case of hacksawing and bandsawing, the tool diameter theoretically assumes an infinite value (translatory cutting movement).

9.1 Turning

Turning is a machining process with a geometrically defined cutting edge, a rotational cutting motion and an arbitrary transverse translatory feed motion [DIN8589a]. For kinematical classification, one always takes into consideration the relative movement between the workpiece and the tool.

Turning methods can be classified from various standpoints. For example different objectives of the machining task lead to the distinction between finish and rough turning. In the case of rough turning, a high material removal rate is reached. In the case of finish turning, the objective is to realize a high level of dimensional

F. Klocke, *Manufacturing Processes 1*, RWTH edition,
DOI 10.1007/978-3-642-11979-8_9,

accuracy and surface quality via small cross-sections of undeformed chip. The flexibility of this manufacturing process allows for economical use from prototype and mass production. In the case of automated and NC operations, several tools can be engaged simultaneously during the machining process in order to reduce manufacturing times and to increase the material removal rate.

The subdivision of turning process variants according to DIN 8589-1 will be presented in the following. Since some of the process variants that appear in the standard are of secondary importance, only the most important process variants will be explained in detail.

Figure 9.2 right shows the cross-section of undeformed chip A. In it,

- b is the width of undeformed chip
- h undeformed chip thickness
- a_p depth of cut
- f feed
- κ_r tool cutting edge angle

Neglecting the inclination, the values can be approximated with the following equations:

$$b \approx \frac{a_p}{\sin \kappa_r} \tag{9.1}$$

$$h \approx f \cdot \sin \kappa_r \tag{9.2}$$

Figure 9.2 and Eq. (9.3) show various calculation possibilities for the nominal cross-section of undeformed chip.

$$A = b \cdot h = a_p \cdot f \tag{9.3}$$

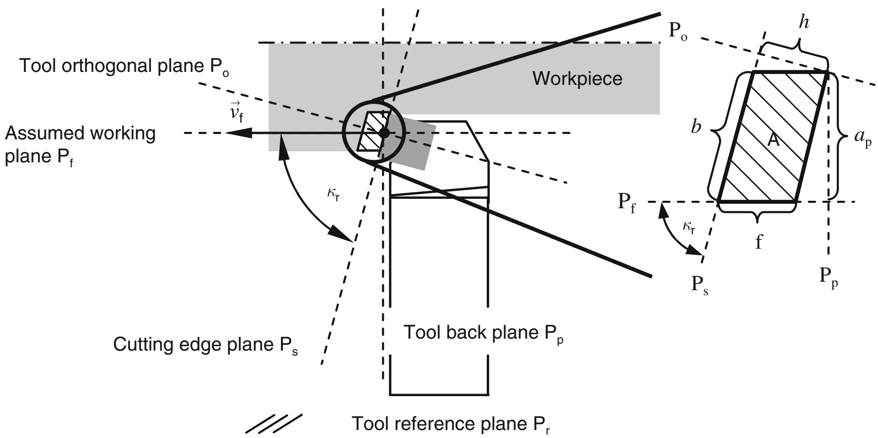


Fig. 9.2 Tool-in-hand system and nominal cross section of undeformed chip

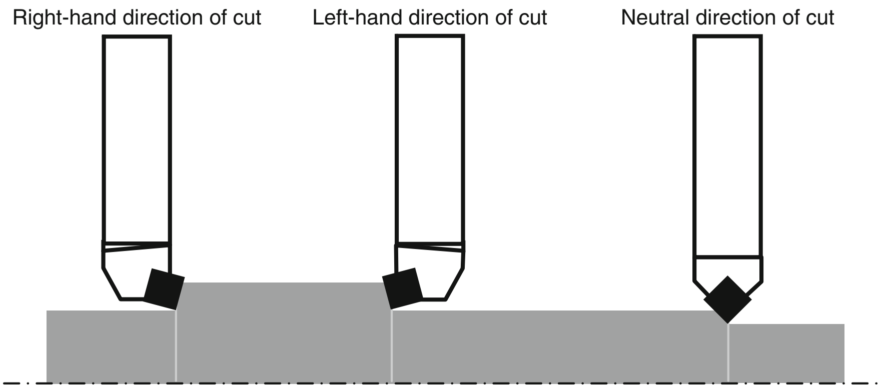


Fig. 9.3 Styles of insert holders

The turning tools of the various process variants are classified analogously to Fig. 9.3 according to the design of their tool holder.

9.1.1 Face Turning

Face turning is a turning method used to produce an even surface orthogonal to the axis of rotation of the workpiece. Process variants include, amongst others, transverse face turning and transverse parting-off for sectioning workpiece components or the entire workpiece [DIN8589a] (Fig. 9.4).

The cutting path of all transverse face turning variants lies on an Archimedean spiral. In the case of cylindrical face turning variants on the other hand, the cutting path is in the shape of a coil (helical line). Face turning operations are usually carried out with automatic lathes, especially in the case of small parts, which are manufactured from a bar. In transverse parting-off operations, the tools are designed to be slender in order to minimize loss of material. Both minor cutting edges are tapered toward the tool shaft in order to avoid jamming. Under heavy strain, the tools tend to clatter due to their geometric design. During face turning processes, one must bear in mind that the cutting speed changes with the tool diameter when machining

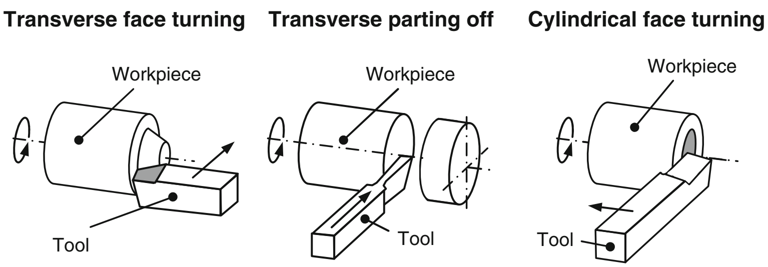


Fig. 9.4 Process variants of face turning, according to DIN 8589-1

with a constant rotation speed. On conventional lathes, a certain cutting speed range is maintained, for example, by multiple, gradual adjustment of the rotation speed to the machining diameter [Degn00]. In the case of lathes with continuous rotation speed control, the cutting speed is kept constant.

9.1.2 Cylindrical Turning

Cylindrical Turning is used to produce a cylindrical surface that is coaxial to the axis of rotation of the workpiece. The use of this method extends from finishing very small parts (e.g. in the clock and watch industry) to heavy roughing forged turbine blades or drive shafts for plant engineering (e.g. cement mills with lengths of up to 20 m).

The most important variants of cylindrical turning are longitudinal cylindrical turning and centreless rough turning (Fig. 9.5). Longitudinal cylindrical turning is the most common method variant, which will be used to exemplify many different machining phenomena as in Chap. 3.

Centreless rough turning is cylindrical turning with several major cutting edges arranged on a rotating tool. The feed movement is made by the workpiece and the rotation movement by the tool. This combination leads to a very high material removal rate. This process variant is predominantly used for removing oxide and roller coatings as well as the surface cracks of rolling and forging blanks such as is required, for example, in the manufacture of cold drawn steel. The surface quality of intermediate products can thereby be improved and impermissible shape deviations avoided. To do this, the minor cutting edge angle κ'_r is kept in the range of $0 < \kappa'_r < 2°$. The depth of cut is generally kept small ($a_p < 1$ mm). The feed is limited by the length of the minor cutting edge and dependent on the demanded surface quality. In steel machining, feeds of up to $f = 15$ mm are used. Surface qualities in the range of $R_{kin} = 2 - 10\,\mu$m can be obtained. Centreless rough turning is much more productive than longitudinal cylindrical turning and reduces the need for subsequent machining due to its high surface quality and dimensional accuracy. Another advantage is that the long rod material need not be guided by steady rests since the rotating tool stabilizes the position of the workpiece, and the protrusion lengths of the workpiece are very short.

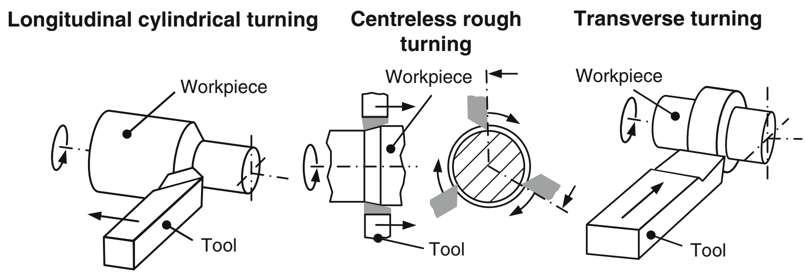


Fig. 9.5 Process variants of cylindrical turning, according to DIN 8589-1

9.1.3 Helical Turning

Helical turning is used to manufacture helical surfaces with profiling tools. Feed corresponds to the pitch of the screw thread. Figure 9.6 shows a few important process variants that fall under this category: thread turning, thread chasing and thread die cutting [DIN8589a].

In the case of thread turning, the thread is manufactured by only one profiled cutting edge in several passes until the required thread depth is obtained. It is characteristic of this process variant that the pitch is produced by the feed. On conventional lathes, the translatory motion is mechanically linked to the rotation motion. In the case of numerically controlled lathes, this link is made electronically.

Thread turning tools are available as both part and full profile tools. Part profile tools can only be used when the workpiece is brought to the required external diameter before thread turning, since only the pitch is cut and the external surface is no longer machined. After thread turning, the depth of the thread must be checked. Full profile tools on the other hand are shaped in such as way that the corresponding thread depth is directly cut from the material so that the output workpiece must not be prepared beforehand (Fig. 9.7).

Thread chasing is similar to thread turning with the exception, shown in Fig. 9.6, that several cutting edges of a tool, offset by the pitch, are engaged simultaneously.

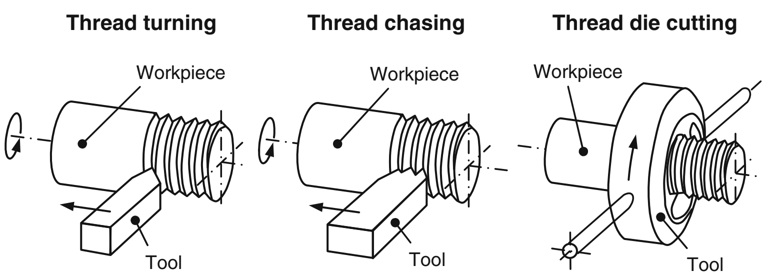


Fig. 9.6 Process variants of helical turning, according to DIN 8589-1

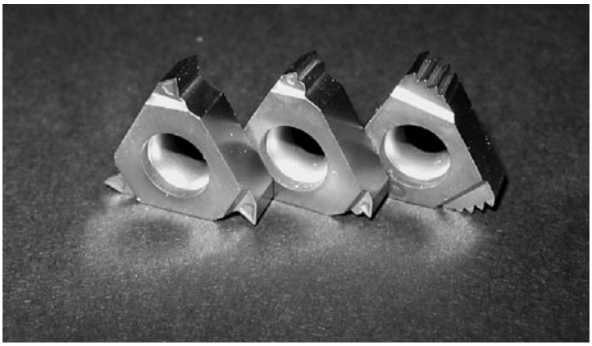


Fig. 9.7 Helical turning: part and full profile tools, chaser

By arranging several profile cutting edges beside each other on one tool, of which every subsequent one is shifted back by the infeed, the thread can be manufactured completely in one pass.

Chasers can be designed as flat or round thread chasers. Round chasers have to be designed as threads themselves so that they do not destroy the manufactured pitch. To chase right-hand threads, a tool with a left-hand thread must be used, and for left-hand threads a tool with a right-hand thread. For internal thread turning chasing, round chasers are usually preferred, as they allow for both a better use of space and a solid tool design. Chasers are also used in die heads, which allow for a radial resetting of the chaser after the thread die cutting process. In this way, it is possible to reset the die head without changing the direction of rotation. We differentiate between three types of thread die heads depending on the type and arrangement of the cutting edges:

- radial chasers,
- tangential chasers and
- round chasers

Chasers are also offered with part profile and full profile. Using a full profile tool makes a higher material removal rate possible. Tool manufacturers sometime also designate chasers as thread turning tools of multi-point design (2–3 teeth as a rule).

Thread die cutting represents a further development of thread chasing – thread chasing with tangentially distributed cutting edges. These modifications alter the process kinematics to the effect that this variant should in fact be considered a helical broaching technique (Chap. 10).

9.1.4 Profile Turning

Profile turning is used to produce rotation-symmetrical workpiece shapes by reproducing the tool profile. Profile turning variants are classified according to their process kinematics. The most common methods, shown in Fig. 9.8, are face profile grooving, transverse profile grooving and transverse profile turning [DIN8589a].

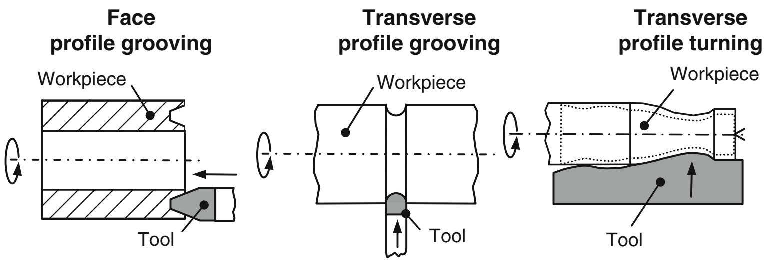


Fig. 9.8 Process variants of profile turning, according to DIN 8589-1

In the case of profile turning, tools made of both high speed steel and cemented carbide are used. Profile tools made of high speed steel are very common, as they are very tough, easy to manufacture and to regrind.

In the case of large cross-sections of undeformed chip and deep profiles, the grooving tools are equipped with chip breakers in order to prevent jamming of the chips in the profile. "Overhead clamping" of the tool can also be beneficial to chip flow. In order to avoid potential clattering during grooving processes due to instabilities in tool clamping, grooves should have a limited width of cut, $b = 15$ mm (in special cases up to 30 mm) and be up to a depth double the size of the chip width (in special cases up to triple the size is possible).

9.1.5 Form Turning

Form turning is used to produce workpiece shapes by controlling the feed movements. Form turning is categorized as in Fig. 9.9 into NC form turning, copy turning and kinematic form turning [DIN8589a].

In NC form turning, the feed movement is realized by electronically linked feed drives. NC form turning is the state of the art today.

Copy turning involves deriving the feed movement from a reference shape, a moulding or a masterpiece. Pure copy turning was developed further when machine tool controls were made available that could store a contour that had once been applied. These are called teach-in processes.

Kinematic form turning was often used in the past to produce ball heads. In this case, the feed axes were kinematically linked via a transmission. This process variant has also been replaced by NC form turning.

Non-circular turning is a special method used to manufacture non-round workpiece surfaces by periodic control of the cutting direction [DIN8589a].

As shown in Fig. 9.10, a distinction is drawn between cylindrical and transverse non-circular turning.

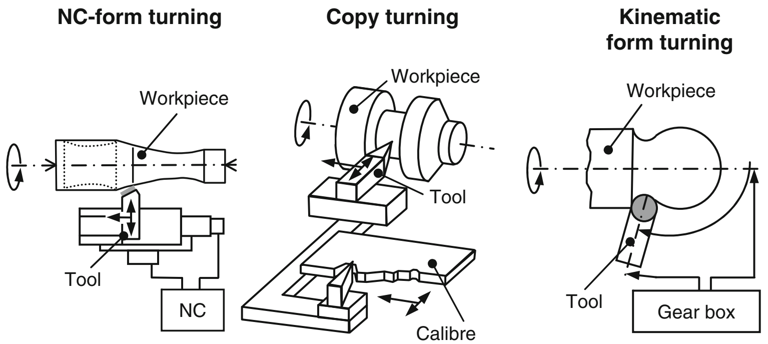


Fig. 9.9 Process variants of form turning, according to DIN 8589-1

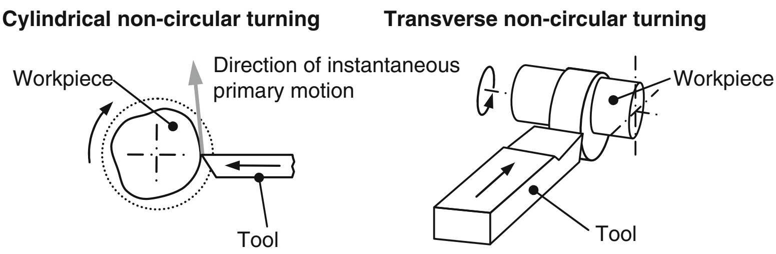


Fig. 9.10 Process variants of non-circular turning, according to DIN 8589a

By control, the turning tool can be advanced with an advancing rotational motion of the workpiece. This can also mean however that the tool is partially no longer being engaged (e.g. square turning). The rotation movement of the workpiece and the feed movement of the tool have a fixed transmission ratio.

9.1.6 Further Process Variants

Up to this point, selected process variants were basically explicated using the example of external machining. In principle, these process variants can also be used for internal machining as shown in Fig. 9.11.

When using internal turning to produce deep contours however, stability problems can arise due to the long protrusion length of internal turning tools. For this reason, the protrusion length and the shaft diameter, which depends on the size of the contour to be machined, should be taken into consideration when selecting the cutting parameters.

Figure 9.12 shows some typical tools used in internal turning.

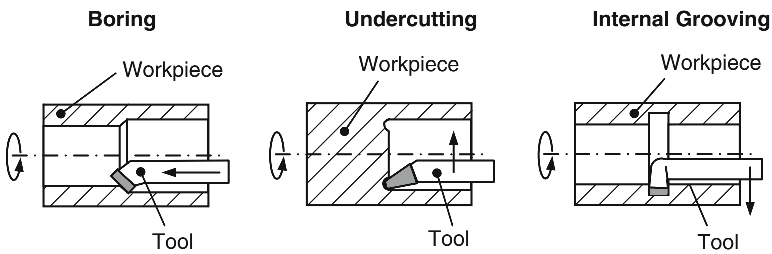


Fig. 9.11 Internal turning, according to DIN 8589-1

9.2 Milling

Milling is a machining production method with a circular cutting movement of a usually multi-tooth tool for producing arbitrary workpiece surfaces. The direction of cut is perpendicular or sometimes transverse to the tool's axis of rotation.

Fig. 9.12 Internal turning: tool design (Source: Sandvik Coromant)

Milling processes are categorized in DIN 8589-3 in accordance with the surface produced, the tool shape (profile) and kinematics as (including others) [DIN8589c]:

- slab milling,
- circular milling,
- hobbing,
- form milling and
- profile milling.

If the workpiece surface is produced by the front face of the tool with the minor cutting edge, it is called face milling (Fig. 9.13). Analogously, milling processes in which the surface is manufactured by the cutting edges on the milling cutter periphery are called peripheral milling (Fig. 9.14).

Depending on the tool's rotation and feed direction, we distinguish further between up and down milling. In order to distinguish up and down milling from each other, Fig. 9.15 shows the movements performed with reference to the workpiece. In practical applications, often the cutting motion is often carried out by the tool and the feed movement by the workpiece.

In order to distinguish both process variants however, it is advisable to relate the movements collectively either to the tool to the workpiece. Down milling is shown in Fig. 9.15, left. At the cutting edge's point of exit, the cutting speed vector v_c and the feed velocity vector v_f point in different directions. The feed direction angle between both velocity vectors is $\varphi = 180°$. In the case of up milling, the cutting speed vector v_c and the feed velocity vector v_f both point in the same direction at the cutting edge exit point. The feed direction angle is $\varphi = 0°$. Depending on the position of the milling tool relative to the workpiece, a milling process can contain elements of both up and down milling, so a clear classification is not always possible. In the case of pure down milling, the cutting edge exits with a undeformed chip thickness of $h = 0\,\text{mm}$, i.e. the minimum undeformed chip thickness is undershot beyond a certain angle of engagement. No definite chip removal occurs at that point,

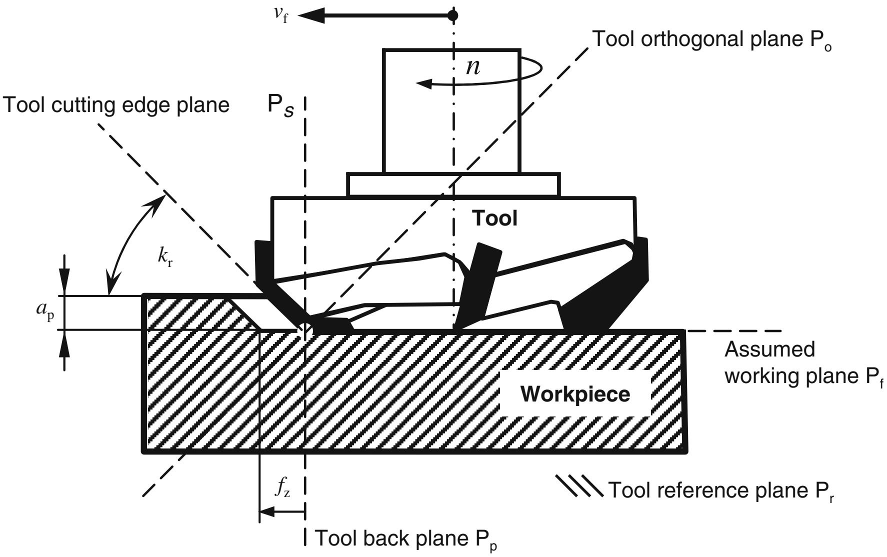


Fig. 9.13 Kinematics of face milling

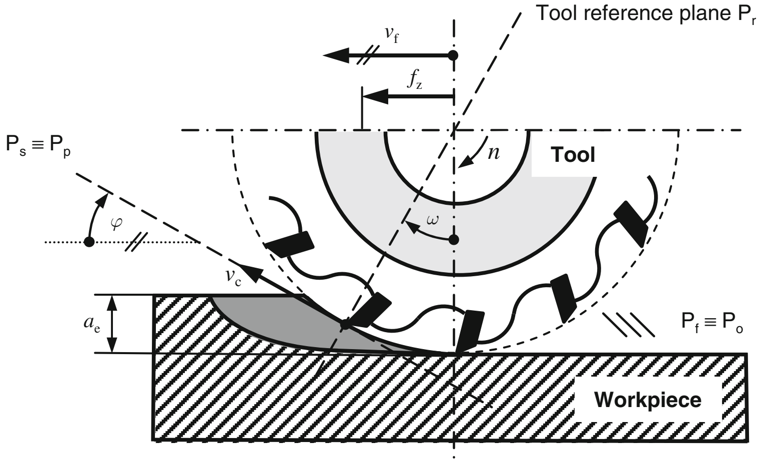


Fig. 9.14 Kinematics of peripheral milling

and solely compressive and frictional processes are occurring. Correspondingly, the cutting edge enters the workpiece with a undeformed chip thickness of $h = 0$ in the case of pure up milling.

The tool orthogonal rake angle γ is composed of a radial component γ_f and an axial component γ_p. As in all other processes, we make a distinction between a positive (γ_f and $\gamma_p > 0°$) and a negative (γ_f and $\gamma_p < 0°$) cutting part geometry.

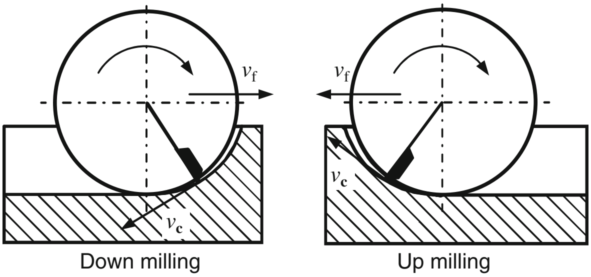


Fig. 9.15 Up and down milling

In all milling processes, the cutting edges, as opposed to other processes like turning or drilling, are not constantly being engaged. Rather, at least one cut interruption occurs per cutting edge during each tool rotation. Because of the constant cut interruptions, the contact conditions between the tool and workpiece are of particular importance for the wear properties of the cutting tool materials in addition to the cutting conditions. Different contacts can result depending on the geometric conditions determined by the milling cutter diameter, size of cut and cutting part geometry (Fig. 9.16).

It is particularly inauspicious if the cutting edge point that is the most sensitive to impact is the first point to make contact with the workpiece. Such "S-contact" can be

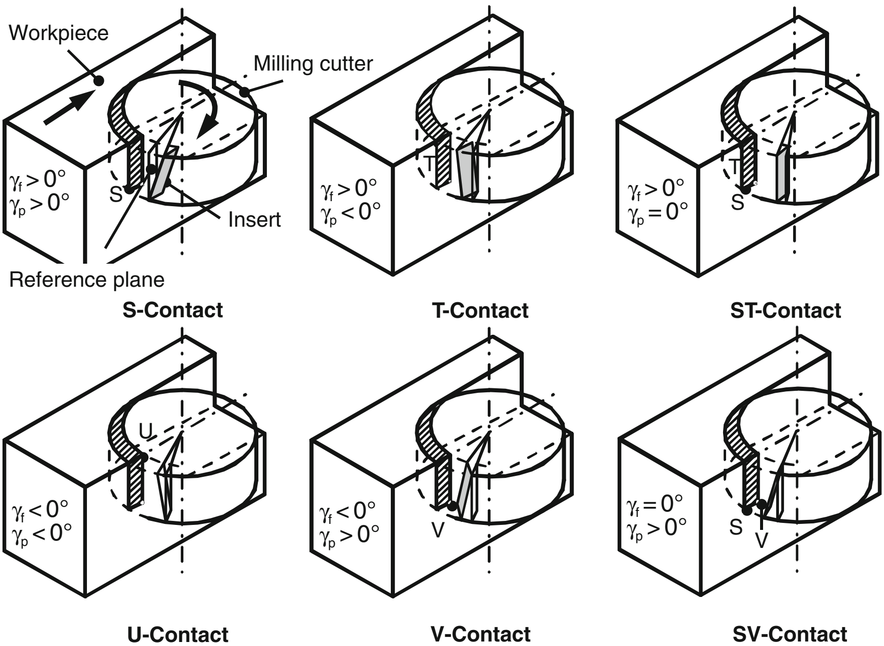


Fig. 9.16 Forms of contact in milling

avoided by varying the cutting edge geometry and feed rate accordingly. The most favourable type of contact is “U-contact”, in which the cutting edge point that is furthest removed from the minor and major cutting edges is the first to make contact with the workpiece. All other types of point or line contact are regarded as intermediate stages between S-contact and U-contact with respect to impact sensitivity [Kron54, Beck69, Damm82].

“Helical chip milling” ($\gamma_f < 0$, $\gamma_p > 0$) has become established as a method for improving chip removal, which is often problematic in the case of rotating tools (Fig. 9.17).

In comparison to the entry conditions, greater importance is attached to the exit conditions with respect to wear caused by fractures [Kron54, Okus63, Hosh65, Beck69, Lola75, Peke78, Peke79, Köll86]. In the case of tool exit with finite undeformed chip thickness (e.g. in up milling), tensile stresses can arise in the cutting edge in the unencumbered state because of resilience, which lead to cutting edge fracture.

This phenomenon must be taken into account when determining milling strategies. The use of tougher cemented carbides may increase the length of tool life until failure, but that alone cannot prevent premature failure in the case of unfavourable exit conditions. In order to improve cutting edge stability, additionally stabilizing protective chamfers are fitted near the corner and on the cutting edge.

Cutting interruptions mean thermal and dynamic alternate stresses for the cutting tool material, which can cause comb and parallel cracks and thus lead to cutting edge fracture. The cutting tool materials used must therefore be very tough, temperature-resistant and have high edge strength [Vier70].

For steel-working, high-speed steel and tough cemented carbides of machining application groups P15 to P40 are used; for machining cast iron, NE metals, plastic and hardened steels types K10 to K30 are used. The cutting tool materials used for milling were developed with an eye to increased thermal and mechanical alternate stress and are thus usually not directly comparable with the cutting tool material types used for turning.

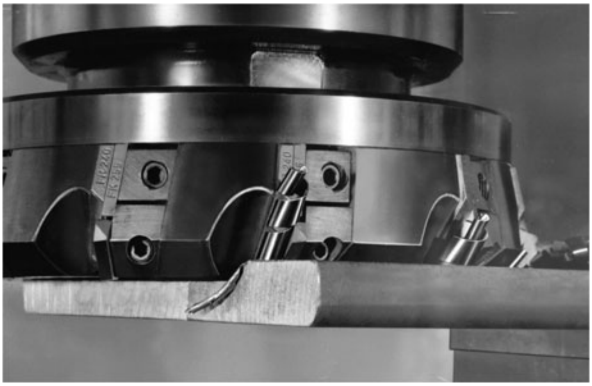


Fig. 9.17 Helical chip milling cutter in use (Source: Walter)

Further developments in cemented carbides and coating technologies have made it currently possible to use coated cemented carbides when milling cast irons as well as steels. When fine milling steels (HB < 300), cermets have also found increasing use. For rough milling grey cast iron, Si_3N_4 ceramics can be used successfully with a high material removal rate. Oxidic and mixed ceramics are suitable cutting tool materials for finish milling grey cast iron, chilled cast iron, case-hardened steels, heat-treated steels and hardened steels, while in the case of hardened or high-strength heat-treated steels (> 45 HRC) PCBN is also suitable. Supereutectic Al alloys, fibre-reinforced plastics and the milling of graphite electrodes for spark erosion are typical applications of PCD-coated tools when used for milling.

9.2.1 Process Variants, Specific Characteristics and Tools

Milling processes are used most frequently to produce level surfaces (linear feed movement: slab milling). Figure 9.18 shows the most important slab milling processes, which differ by their kinematics and engagement conditions, and designates the slab milling tools associated with them. In practice, milling processes are usually named according to the type and form of milling tools used, e.g. plain milling, end milling, side and face milling, face milling, profile milling etc.

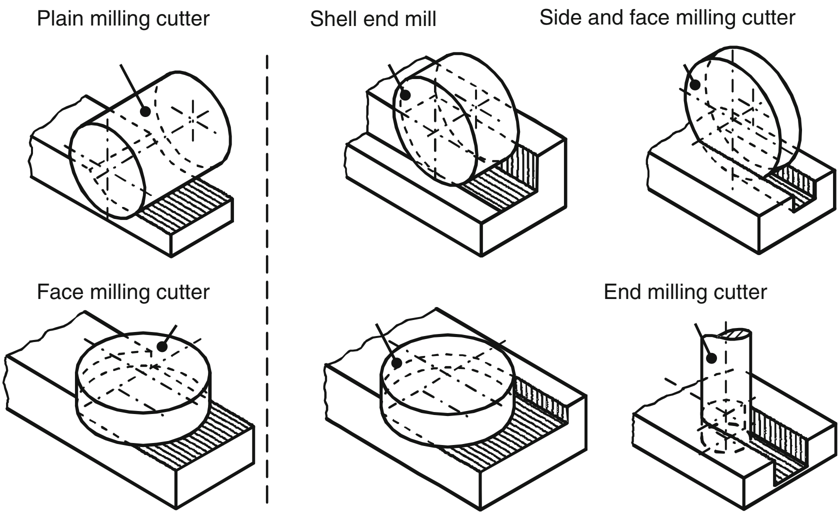


Fig. 9.18 Slab milling processes

9.2.1.1 Face Milling

In the case of face milling, the width of cut a_e is much larger than the depth of cut a_p, and the workpiece surface is created by the minor cutting edge. If the lead angle is $\kappa_r = 90°$, this milling process is also referred to as edge milling. In this case, the workpiece surface is created with both the minor and major cutting edges.

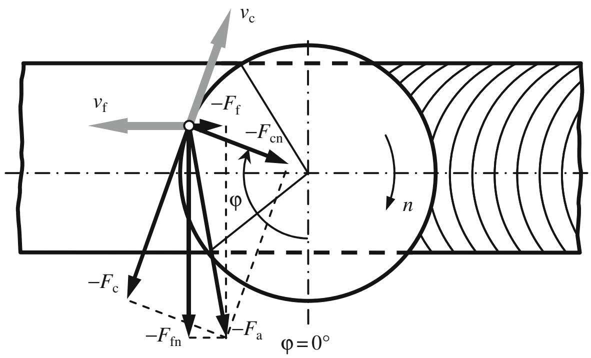


Fig. 9.19 Cutting force components at head face milling, according to KAMM [Kamm77]

According to DIN 6584, the resultant force F involved in milling can be broken down into an active force F_a on the working plane and a passive force F_p perpendicular to the working plane (see also Sect. 3.8.1). The direction of the active force F_a depends on the feed direction angle φ. The components of the active force can be related to the direction of the cutting speed v_c (cutting force F_c and cutting normal force F_{cn}) or to the direction of the feed velocity v_f (feed force F_f and feed normal force F_{fn}) as in Fig. 9.19.

KIENZLE's resultant force equation [Kien52] can also be applied to milling. For the components of the resultant force F – cutting force F_c, cutting normal force F_{cn} and passive force F_p, we have:

$$|\vec{F}_i| = k_i \cdot b \cdot |\vec{h}|^{1-m_i} \tag{9.4}$$

with:

$$i = c, cn \text{ and } p$$

Due to the wide range of undeformed chip thicknesses covered by milling, the KIENZLE relation is only valid for certain areas. The range of undeformed chip thickness of $0.001 < h < 1.0$ mm is subdivided into three sections (Fig. 9.20). For each range, a line can be ascertained, which is defined by the specific resultant force k_i and the exponent m_i:

$$|k_i| = k_{i1.0,01} \cdot |\vec{h}|^{-m_{i0,01}} \text{ for } 0.001 < h < 0.01\,\text{mm}$$

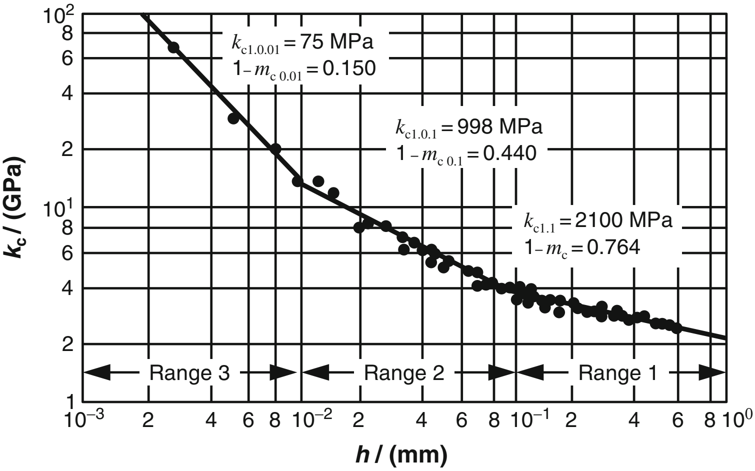
$$|k_i| = k_{i1.0,1} \cdot |\vec{h}|^{-m_{i0,1}} \text{ for } 0.01 < h < 0.1\,\text{mm}$$

$$|k_i| = k_{i1.1} \cdot |\vec{h}|^{-m_i} \text{ for } 0.1 < h < 1.0\,\text{mm} \tag{9.5}$$

with:

$$i = c, cn \text{ and } p$$

As long as the characteristic values are known for the existing marginal conditions (material, cutting tool material and cutting conditions), the respective resultant



Material: Ck45N
Cutting tool material: HW-P25
$v_c = 190$ m/min

γ_f	γ_p	α_f	α_p	λ_s	κ_r	ε_r
–4°	–7°	6°	23°	–6°	75°	90°

Fig. 9.20 Specific cutting force in face slab milling [Kamm77]

force component F_i can be calculated for milling. To estimate the resultant force however, characteristic values are often utilized that have been established in turning processes.

To mill very small, level and right-angled surfaces, grooves with square cross-sections and long slots, solid face milling cutters made of HSS or cemented carbide are used, beyond a tool diameter of $D = 10-16$ mm tools with clamped cemented carbide indexable inserts and in case of increased requirements on surface quality, dimensional accuracy and performance milling cutters with soldered cemented carbide cutting edges [Sack76].

The size and number of teeth of the milling tool are selected based on the dimensions of the workpiece surface to be machined and the drive capacity of the machine. The tooth pitch of the tool depends on the form and size of the tool, on the available machine power and on the chip formation of the material. Short-breaking chips require a small chip space and thus a small pitch. Large cutter heads for cast iron machining can thus be fitted with up to 200 inserts. In order to prevent chattering of the tool/workpiece/machine system with the frequency of the cutting edge engagement, face milling heads are partly manufactured with an uneven pitch on the periphery.

Face milling heads up to a diameter of $D = 250$ mm are mounted in the usual fashion on the tool spindle. Larger face milling cutters are designed in two parts because of their great weight for better handling during tool change. The base body remains on the spindle during tool change, so only the ring with the clamped cutting edges is changed.

Fig. 9.21 Face milling head with tool cartridges (Source: Kennametal, Hertel)

Another potential way to increase efficiency and universality is the use of face milling heads with tool cartridges (Fig. 9.21). Depending on the requirements, tool cartridges can be inserted in a base body that can receive various kinds of indexable insert (three-corner, four-corner, round), sizes and geometries (e.g. positive, negative, $\kappa_r = 90°$, $\kappa_r = 75°$, with moulded chip breakers etc.).

In general, steel-working employs positive cutting part geometries and, to improve chip removal, helical chip geometries. In the case of welded constructions or larger material inhomogeneities, a negative cutting part geometry is more advantageous for preventing cutting edge fracture. The same is true for machining materials with high strength and toughness. Milling with cutting ceramics is generally performed with a negative cutting part geometry.

In order to avoid re-cutting the face milling cutter due to elastic form changes in the overall system, the milling axis can be tilted by 0.5–1°. However, this sets the feed direction.

The tool cutting edge angle amounts to $\kappa_r = 45\text{–}75°$ in face milling (special case: corner milling $\kappa_r = 90°$). It affects to a large extent the size of the active and passive forces and thus the stability of the milling process, especially when machining thin-walled parts (e.g. welded gearboxes) or in milling operations on milling and boring machines with a widely projecting spindle.

Generally, the face milling cutting conditions are selected lower than in turning processes. Smaller cross-sections of undeformed chip are chosen in particular in order to keep the dynamic stress on the cutting tool materials low and to prevent tool fracture.

Face milling is used both for pre-machining and also increasingly for finishing purposes. Finishing with geometrically defined cutting edges is becoming increasingly important because of the potential of single-machine processing. Finish face milling is especially used as a finishing process for large even surfaces with special surface quality and smoothness requirements when other finishing methods (e.g. grinding or shaving) are uneconomical or impossible. Such machining problems usually arise in heavy machine construction, e.g. when producing joining surfaces, machine tables and guideways on machine tools and when milling sealing faces in engine and turbine construction.

Indexable inserts for finishing have an active minor cutting edge, i.e. in these tools the cutting edge angle is $\kappa'_r = 0°$, so the chamfer of the minor cutting edge lies parallel to the workpiece surface. The chamfer length of the minor cutting edge is generally $L'_{sa} = 2-3$ mm, in the case of special wide finish milling tools $L'_{sa} = 10-15$ mm. The feed per tooth should not exceed 2/3 of the length of the active minor cutting edge nad. There are three kinds of finishing tools (Fig. 9.22):

- Conventional finishing face milling cutter which work with small depths of cut and feeds per tooth and are fitted with a large number of teeth.
- Wide finishing face milling cutters, which are equipped with a small amount of teeth (1–5) and work with very small depths of cut and high feeds (Fig. 9.23). In the case of such tools, the minor cutting edge nads are equipped with large radii to simplify tool pre-adjustment. This helps to obtain a very good surface quality

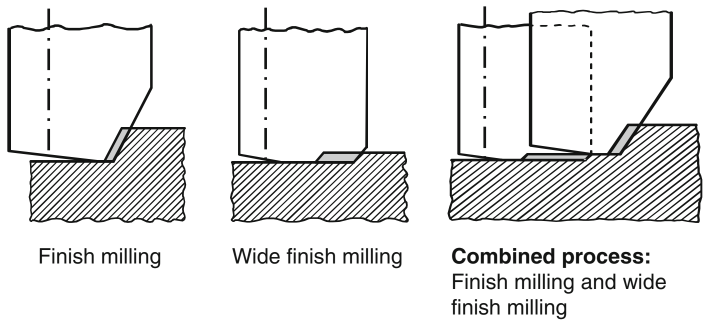


Fig. 9.22 Processes for fine milling (Source: Siemens)

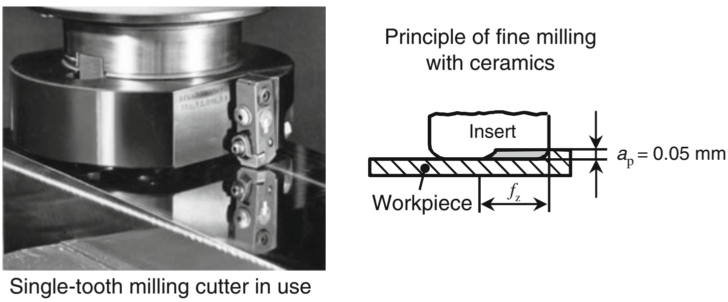


Fig. 9.23 Wide finish milling (fine milling) with ceramics (Source: Feldmühle)

in cutting, similarly to shell turning. However, the jerk forces are larger than in conventional finishing face milling, which can result in an axial displacement of the tool. Ceramics are the predominate cutting tool materials in this case.

- Face milling with finishing cutting edges and wide finishing cutting edges that combine the advantages of both methods. The tool is in this case only fitted with one or two wide finishing cutting edges that are radially set back and that protrude axially by 0.03–0.05 mm to produce a high surface quality. The width of the finishing cutting edges should correspond to about one and one half times the feed per rotation.

For finishing, the pre-adjustment of the cutting edge has increased significance. If no special, finely adjustable tool holding fixtures are used, all finishing face milling tools should be ground and lapped prior to use on the machine tool and after fitting with the cutting edges in order to obtain the necessary face and run-out accuracy (< 5 μm) corresponding to the required workpiece surface quality. Faulty insert pre-adjustment increases kinematic roughness and shortens the tool life sometimes considerably.

The cutting speed is selected high in finishing (e.g. up to $v_c = 300\,\mathrm{m/min}$ in steel-working with cemented carbide) in order to obtain a high surface quality. In the case of steel, surface finishes of $R_t = 5\text{–}10\,\mu\mathrm{m}$ are obtained, in the case of grey cast iron $R_t = 1\text{–}5\,\mu\mathrm{m}$.

9.2.1.2 Peripheral Milling

In the case of peripheral milling, the workpiece surface is created by the major cutting edge. There is peripheral down milling and peripheral up milling. In peripheral down milling, the cutting force acts upon the workpiece (Fig. 9.24), while in peripheral up milling it is directed away from the workpiece so that an unstable workpiece (e.g. a thin steel sheet) can be lifted from the clamping surface or be induced to chattering.

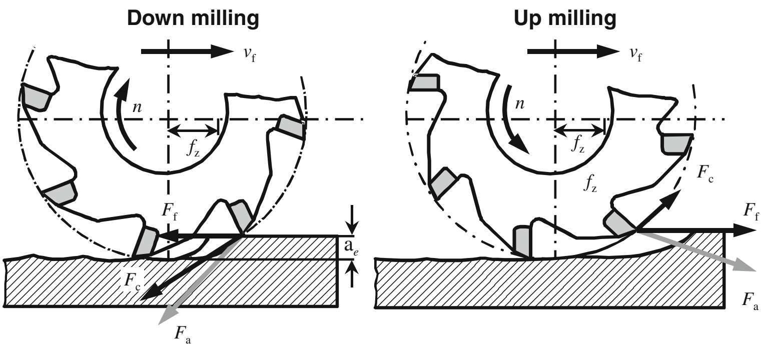


Fig. 9.24 Peripheral up- and down milling

In the case of peripheral down milling, a table feed drive that is free of play is necessary in order to prevent vibrations and impacts. While in the case of peripheral down milling the lead takes place with an approximately full cross-section of undeformed chip, in up milling the cross-section of undeformed chip is slowly increased. This can lead to material compression and thus to the formation of a poor surface.

Besides the usual HSS tools, cemented carbide peripheral milling cutters or plain milling cutters are finding increasing use. When the cutting edges have a coaxial adjustment, high dynamic stresses come into play because one whole cutting edge enters into or exits from the material at a time. In the case of helically toothed tools, the dynamic load can be reduced, but an axial force arises then which can lead to tool or workpiece displacement. Pitch-induced axial forces can be compensated by mutually bracing a right-inclined and a left-inclined plain milling cutter of the same design (Fig. 9.25).

This disadvantage can be overcome with a double helical gearing with opposite pitch. Such tools are very expensive both to acquire and to prepare however (Fig. 9.25).

Should sharp-edged profiles with good dimensional and formal accuracy be prepared, combined peripheral face milling cutters or plain milling cutters are used (Fig. 9.26), which are relief-ground on the front face of all cutting edges (formation of a tool orthogonal clearance).

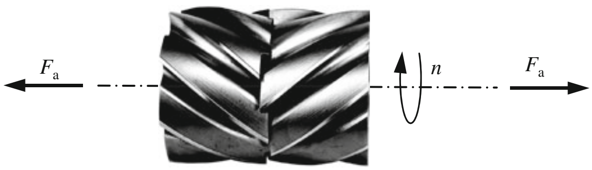


Fig. 9.25 Combined plain milling cutter with opposite hand helix

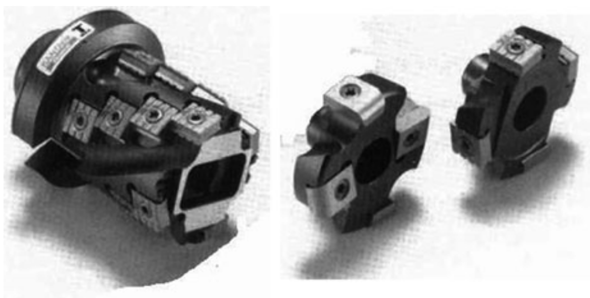


Fig. 9.26 Modular shell end mill (Source: Sandvik Coromant)

9.2.1.3 End Milling

End milling is a continuous peripheral face milling process which uses an end milling cutter. This process is advantageous when manufacturing mould surfaces

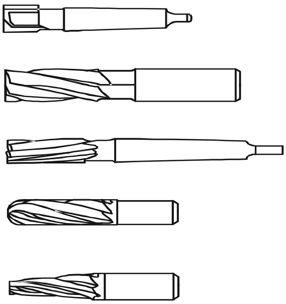


Fig. 9.27 End milling cutter

(e.g. in die construction) as well as forming grooves, pockets, slots and cavities of all kinds and sizes.

End milling cutters have to be designed in many cases with a large degree of slenderness ($l/D > 5$–10) depending on the application (e.g. milling deep engravings in dies and moulds). This causes on the one hand, depending on the contact and engagement conditions, chatter vibrations during the process, which can lead to increased wear via fracture, especially in the case of hard, brittle cutting tool materials. Additionally, both chattering and bending of slender tools lead to dimensional and shape inaccuracies in the components. Measures taken to avoid these phenomena should be sought in an optimization of the tool and cutting part geometry, engagement conditions and milling strategy as well as of the cutting conditions [Schr74, Köni80, Hann83, Köll86].

End mills correspond to shell end mills in their construction; for clamping, they are equipped with a parallel shank (with side-clamping and/or fastening thread) or with a taper shank (Morse taper or steep-angle taper; sometimes with fastening thread).

A distinction is drawn between right-cutting and left-cutting tools as well as between right-hand spiral, left-hand spiral and straight-toothed tools (Fig. 9.27). The mill form can be designed cylindrically, conically or as a custom design depending on the machining task. The front face of the tool is generally round or half-round; in the case of tools capable of drilling the face cutting edges must reach as far as the tool centre.

HSS end milling cutters are classified into tool applications groups in accordance with DIN 1836 depending on the material to be machined (Fig. 9.28).

Profiling of the cutting edges in the case of roughing tools leads to a division of the chips into smaller chips. The advantages of these chip dividers include improved chip removal and cutting fluid access as well as reduced stress on the cutting edges (Fig. 9.29).

The design of individual milling cutter geometries and chip divider forms differ depending on the manufacturer.

In principle, all the cutting tool materials are potentially applicable in end milling, depending on the selection criteria regarding workpiece materials and

Group of application	Field of application	Tool
N	Machining of materials with normal strength and hardness	
H	Machining of hard, hard tough and/ or short-chipping materials	
W	Machining of soft tough and/ or long-chipping materials	

Fig. 9.28 Tool application groups

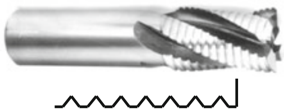
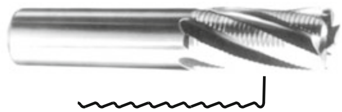
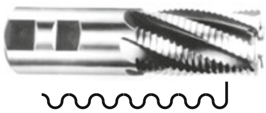
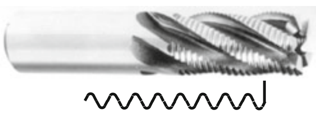
Profiled cuttting edges		
Profile	Group N	Group H
Flat profile (F)		
Rounded profile (R)		

Fig. 9.29 Cutting profiles of roughing-end milling cutters (Source: Fette)

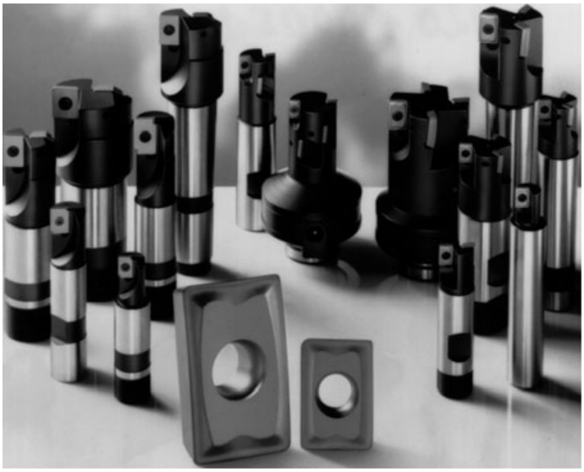
stability. High speed steel is still predominately used – often coated – as well as cemented carbides. Besides solid steel tools, tools with soldered cutting edges and clamped or bolted indexable inserts are used (Fig. 9.30).

A more flexible adjustment of the cutting tool material to the machining task is possible by using tools with indexable inserts. Especially mentionable in this context is the use of different cutting tool materials in a single tool. This can be advantageous, for example, in the case of ball path milling cutters, which are subject to highly diverse stresses along the cutting edges.

9.2.1.4 Profile Milling

Profile milling is milling with forming tools to produce profiled surfaces, e.g. for milling grooves, radii, gear wheels and gear racks as well as guideways.

Fig. 9.30 End milling cutter with inserts (Source: Sandvik Coromant)



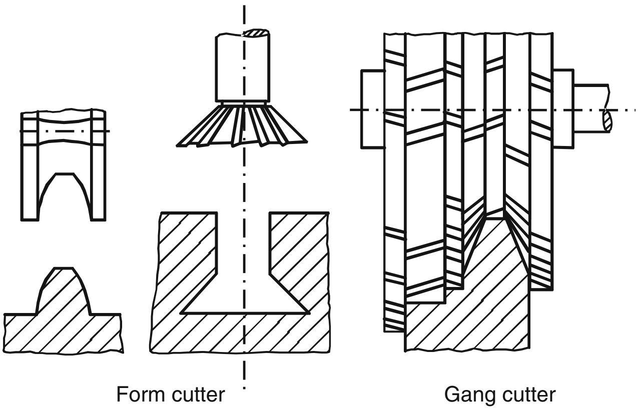


Fig. 9.31 Profile milling cutter

Profile milling tools are adjusted to the form of the profile to be produced. In most cases, a peripheral face milling process is the result. As shown in Fig. 9.31, the tools are designed in one part (form milling cutters) or in multiple parts (gang milling cutters).

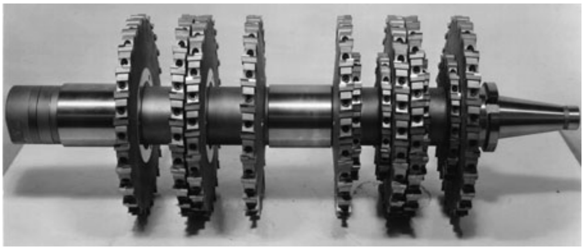
Profile milling cutters are in many cases manufactured as solid HSS tools due to the favourable machinability and inexpensive price. Increasingly however, cemented carbide indexable inserts or soldered cemented carbide cutting edges are being used (Figs. 9.32 and 9.33).

9.2.1.5 Hobbing

In almost all areas of technology, gears are employed as components of an exact and effective transmission of motion. For high-precision gears, the gears are

Fig. 9.32 Gang cutter for machining of grey iron profiles (Source: Walter)

Fig. 9.33 Gang cutter for machining of machine beds (Source: Walter)



manufactured primarily by machining. Because of its high efficiency, hobbing is the dominant machining process for producing externally toothed cylindrical gears.

In hobbing, the coupling of a worm with a worm gear is simulated, whereby a worm interrupted by gashes represents the tool and the worm gear represents the workpiece to be manufactured.

The kinematics of the process will be explained briefly with the help of Fig. 9.34. The rotary movements of the hob and the gear serve to remove the chips. Depending on the hobbing method, superimposed over these are translatory motions of the tool in the axial and tangential directions as well as in the radial direction in order to reach the depth of cutting. This results in the hobbing methods axial, radial-axial,

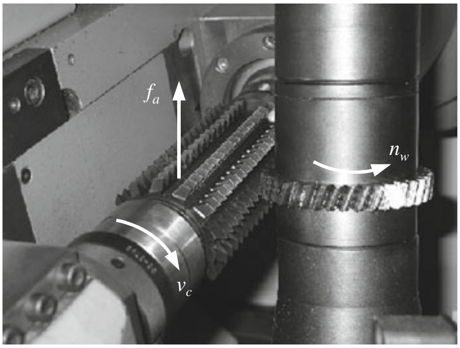


Fig. 9.34 Kinematics of hobbing

tangential, and diagonal hobbing. In industrial production, axial hobbing is currently the most commonly employed process.

The feed in one direction is defined as the distance travelled per workpiece rotation. We distinguish between climb and conventional cutting by means of the direction of the axial feed f_a. In climb cutting, the cutting speed and axial feed motion are directed opposite relative to the workpiece. For wear-related reasons, climb cutting is generally preferred, especially in dry hobbing. The traverse path of the hob can be subdivided into the tool inlet and outlet phases as well as that of full cut, where only in full cut does one obtain the theoretical maximum circular cut lengths and material removal rates (Fig. 9.35).

There are two further process variants used to machine helical gearings, hobbing in the same direction and in the opposite direction, whereby the pitches of the tool and working gear are aligned in the same direction or in opposite directions.

In order to distribute the tool wear arising during the process evenly along the hob for more a efficient use of the tool, the hob can be shifted along its axis in discrete steps. This displacement takes place as a rule after each machined workpiece and is also called "shifting".

The parameters of a hob can be seen in Fig. 9.35, which shows the tool in the machining sequence. The hob is a cylindrical screw interrupted by chip flutes which bring about the gash. The number of worm threads on the cylinder determines the thread number of the hob. The hob teeth are shaped so that tool orthogonal clearances and the potential of regrinding the rake face without altering the tooth profile are created. The pivoting angle or lead angle of the hob results from the direction and size of the helix angle and the pitch angle of the hob worm.

Hobs are classified into three different groups with reference to their construction type (Fig. 9.36).

Solid steel hobs are manufactured from solid material, whereby the entire body must be constructed from high-quality HSS or cemented carbide. Inserted blade hob and cutters with indexable inserts on the other hand consist of a base body made of a more inexpensive material. These hobs are especially suited to manufacturing gears

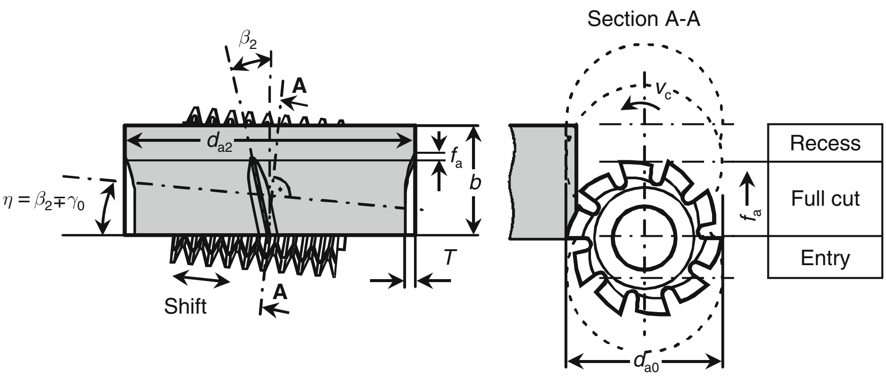


Fig. 9.35 Terms of the hobbing process

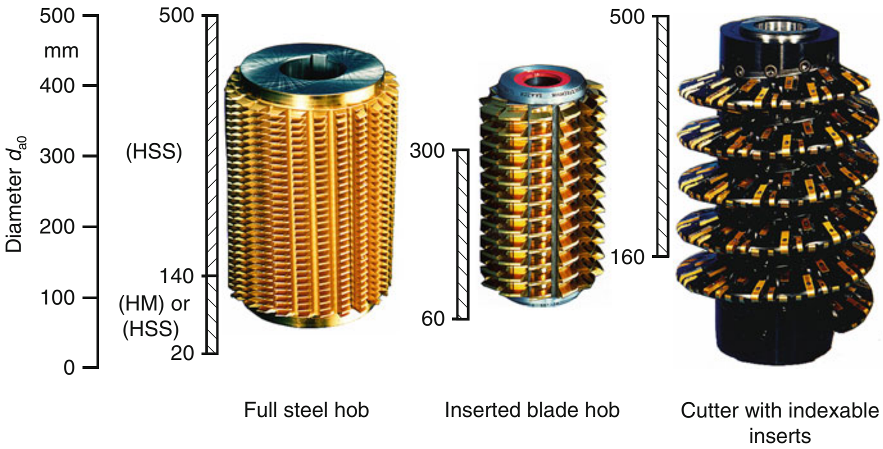


Fig. 9.36 Hob types (Source: Fette, Saacke, Saazor)

with large diameters and large modules. Besides the potential of realizing larger constructive tool orthogonal clearances, the inserted blade hob also has a relatively large potential usable tooth thickness (large number of regrinds). Fastening is done with lateral clamp rings. In the case of cutters with indexable inserts, the tool cutting edge, which is made of cemented carbide, is used upto four times and not reground. This hob type is only suitable for workpieces beyond a modulus of 5 mm because of the resulting poorer cutting quality.

With respect to the selection of the substrate/coating system, a sufficient amount of substrate toughness is of especial importance in hobbing because of the interrupted cut. At the same time, the tool must have as much wear resistance as possible against abrasive and thermal wear mechanisms in conjunction with a suitable hard material coating. By the establishment of (Ti,Al)N coatings is it possible to perform dry machining processes at cutting speeds up to $v_c = 200\,\mathrm{m/min}$, even with tool systems based on HSS [Wink05].

More and more, coated gear tools are reconditioned. The coating is removed, the tool reground and finally recoated on the rake face after the tool operating life is reached [Klei03]. In this way, we can work with consistently high cutting speeds in all tool cycles, and the tool's operating life can be held constant. For reasons of accuracy, ground cutters cannot be over-coated more than 5–10 times because of the coating application on the already coated rake face.

Figure 9.37 clarifies the hobbing process by looking at the creation of a tooth gap. Due to the process kinematics and the resulting shifting between the tool and workpiece, the material of a tooth gap is machined in the successive engagements (hobbing positions) of the individual teeth of a hob thread as can be seen in the sketch in the bottom left of the illustration. The evolvents on the tooth of the workpiece are approximated by profiling cuts. Every cutting tooth makes a cut after one tool rotation in a further tooth gap determined by the hob thread number, but in the same hobbing position; i.e. it is always removed one chip with the same cross-section.

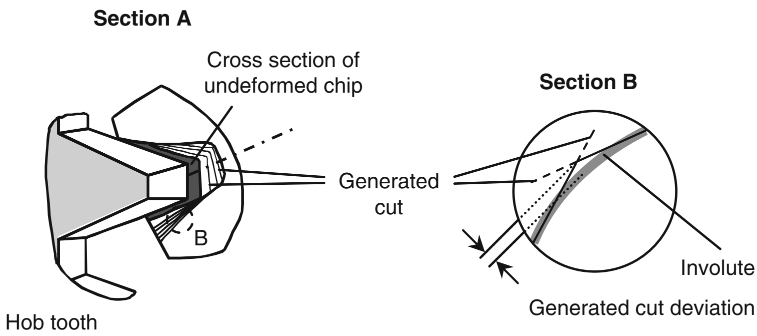


Fig. 9.37 Ratio of engagement in the hobbing process

Due to the differing penetrations between the hob and the workpiece in the particular hobbing positions, variously thick and variously formed chips result (Fig. 9.38). During the first tooth engagements, a large amount of the gap volume is machined, so that here, especially shortly before the middle position, exist the largest cross-sections of undeformed chips. In the following hobbing positions, the tooth gap is mostly profiled and the cross-sections of undeformed chip are reduced. The penetration areas are highlighted in the image.

During hobbing, as in other gears manufacturing processes, changes to the structurally given tool orthogonal clearance and tool orthogonal rake angles occur during the cutting process [Sand72, Sulz73, Sulz74, Köni79]. Figure 9.39 illuminates the cause of this.

At the cutting point under consideration on a certain hobbing position, the effective cutting speed v_r results from the cutting speed of the hob v_c and the hobbing speed v_A. On the entering tool cutting edge (Fig. 9.39), this leads to an enlargement of the structurally given tool orthogonal clearance but simultaneously to a reduction of the tool orthogonal rake angle. This means that, during the cutting process, the

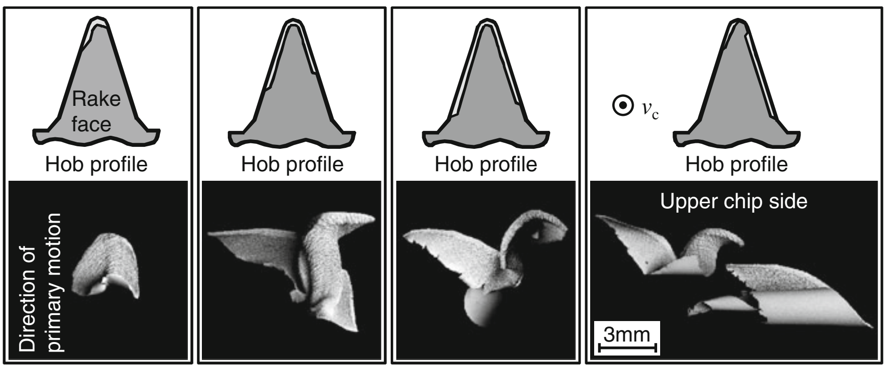


Fig. 9.38 Chip formation in the hobbing process

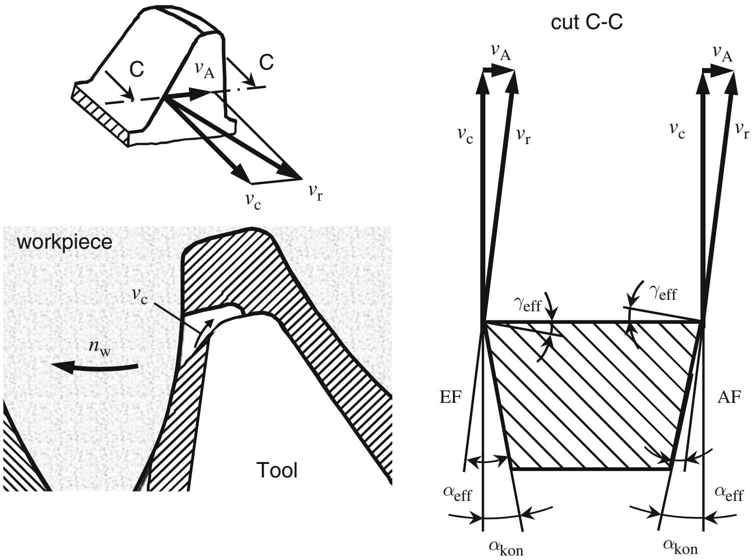


Fig. 9.39 Relative velocity and effective cutting geometry

effective tool orthogonal clearance is larger on the entering face than the structurally given one. The effective tool orthogonal rake angle, on the other hand, is smaller. There are different ratios on the exiting face, upon which the effective tool orthogonal clearance is small and the effective tool orthogonal rake angle is larger than the corresponding structurally determined angles.

A penetration calculation can be applied to simulate the cutting process [Wink05]. In the penetration calculation, the workpiece is analyzed into a certain number of parallel planes. By recreating the axis motions of the hobbing machine, the cutting path of the hob teeth are generated so that they cut the workpiece planes. From the intersection we can calculate a penetration range that corresponds to the chip geometry of the respective hobbing position in the process [Weck02, Weck03].

Figure 9.40 shows a simulated chip geometry in a certain hobbing position. The 3D chip diagram reproduces the distribution of cross-sections of undeformed chips along the uncoiled cutting edge along the cutting arc, whereby the beginning of the cut is in the foreground. The calculated chip geometries form the basic data of a simulation-supported assessment of the stresses affecting the hob during the machining process. It is clear that the chip thickness (top chip thickness) in the top region of the hob can be much larger than in the edge region.

In addition to cemented carbides, PM-HSS cutting tool materials have also become established as substrate materials for hobs in dry hobbing processes. Since both cutting tool materials are in competition with one another, Fig. 9.41 shows the advantages and disadvantages of both tool systems. The advantages are designated with bright points, the disadvantages by dark points.

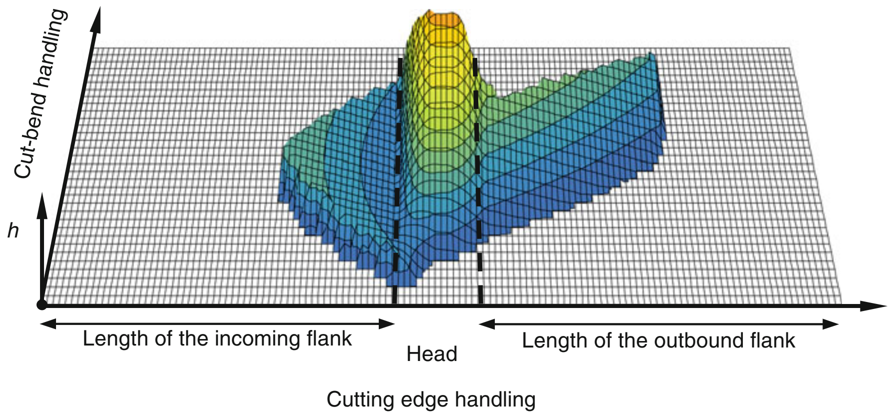


Fig. 9.40 Tension geometry in hobbing

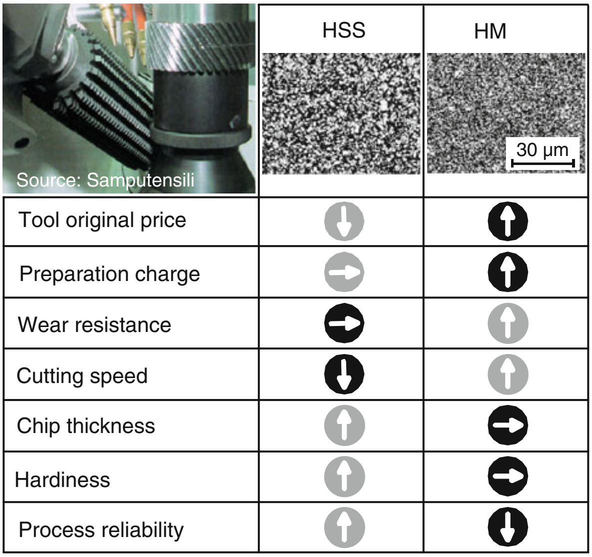


Fig. 9.41 Cutting tool materials for dry hobs

One of the main disadvantages of cemented carbide hobs in comparison to the HSS variant is the higher cost of acquiring the tools. They are generally at least three times more expensive. Because of the higher tool costs, it is only economical to use them if a significantly higher productivity or longer tool life is obtainable.

Because of the much higher wear resistance of cemented carbide, cemented carbide tools can realize clearly higher cutting speeds. However, PM-HSS hobs can realize larger maximum head chip thicknesses. Given the high tool price, the use of cemented carbide hobs is uneconomical in this case, provided the tool life is not significantly higher.

In large batch production, process safety is particularly important. While HSS hobs, not least because of their high toughness, make a relatively high level process safety when their wear behaviour is monitored [Coop99, Kölk99], cutting edge fractures occur occasionally in the case of cemented carbide hobs [Kloc99a, Sulz00]. If this occurs only sporadically, such fractures cannot be detected with process monitoring systems and lead to increased tool wear. This problematizes an economical use of cemented carbide hobs given the multiple reconditioning of the tool that is required. Although dry hobbing used to always be performed with cemented carbide tools, PM-HSS tools have become continuously more popular in the last several years because of their superior toughness properties and lower tool costs [Coop99].

9.2.1.6 Skive Hobbing

Skive hobbing is a continuous process using geometrically defined cutting edges for machining pre-milled gear teeth in a hardened state. Primarily, deformation caused by the heat treatment is removed and the surface quality improved. The process kinematics are identical to those of hobbing.

The concept of skive hobbing is derived from the "peeling cut". Hard finishing with geometrically defined cutting edges requires that small cross-sections of undeformed chip are selected. The skive hobs are designed with a negative top tool orthogonal rake angle, which acts as a negative inclination angle on the tooth flanks. This guarantees that the first contact between the cutting tooth and the workpiece surface is not made directly on the cutting edge but in the stable cutting part area behind it (see also contact conditions, Fig. 9.16).

In order to execute a skive hobbing operation, the tooth gaps of the pre-milled gear teeth must be prepared to such an extent that the top of the skive hob does not engage and only the flank cutting edge is cut. Otherwise, the danger of fracture is increased [Faul86]. The tooth base can be free milled in two different ways: by pre-milling with tools corresponding to reference profile II acc. to DIN 3972 or by pre-milling with protuberance.

In case 1 the pre-processing of the gaps is done with hobs of reference profile III, a sharp edge appears in the tooth base after skive hobbing, which can have a negative effect on tooth base strength. On the other hand, the tooth gaps pre-milled with protuberance have a rounded transition in the tooth base after skive hobbing (Fig. 9.42).

In skive hobbing, it is very important for the sake of consistent cutter wear and the output that the hob tooth is exactly positioned or "centred" in the tooth gap. After centring, the same amount of material is removed on both workpiece flanks (right and left flanks) in the ideal machining case. Centring is made more difficult by the fact that the flank allowance fluctuates along the workpiece periphery and the tooth width due to pre-gear cutting deficiencies, faulty alignment for skive hobbing or as a result of deformation due to hardening.

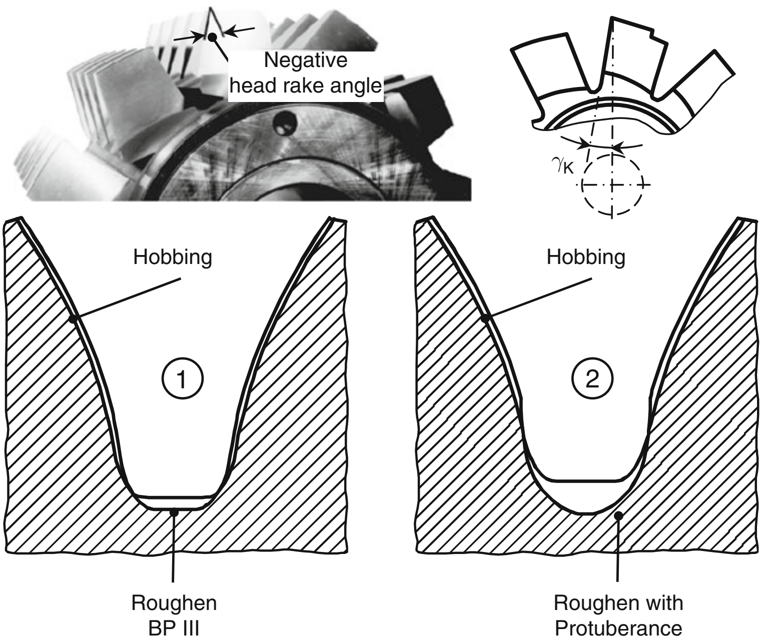


Fig. 9.42 Gap profiles of different rough cutters

Although only very thin chips are removed during the process, very high resultant forces are in play, which are inconstant – especially in the entrance and exit areas – and also change their signs. For this reason, a high static and dynamic stiffness must be required of the hobbing machine in addition to geometric and kinematic accuracy [Faul86].

In the case of skive hobbing, the obtainable length of tool life is depends greatly on the cutting edge, workpiece geometry, cutting parameters and the hardness of the workpiece to be machined. Favourable wear behaviour is exhibited by the ultrafine-grain cemented carbides of ISO application range K10 to K20 in conjunction with a hard material coating [Kais92, Köni95].

Skive hobbing is used as a finishing process or can also serve as a preparation process for a subsequent grinding operation. The grinding costs are thereby levelled by lowering the amount of deformation due to hardening prior to the subsequent hob grinding process. This production sequence is especially of interest for large-module workpieces (larger module 10 mm).

The limits of quality in finishing are basically determined by the feed marks and profiling cut deviations (Fig. 9.37) characteristic of hobbing. With the help of a honing operation following the skive hobbing process, they can be removed at least in the case of small gears. Furthermore, gears that cannot be ground because of their geometry (large grinding wheel diameters) are also made by skive hobbing [Koep94].

9.2.1.7 Turn Milling

Turn milling is a machining process in which the principles of turning and milling are combined in such a way that (generally) rotation-symmetric workpieces are machined with an inserted-tooth cutter on a rotating workpiece. The self-propelled cutter works like a turning tool with a longitudinal feed parallel to the workpiece.

In principle, there are two process variants in turn milling. If the tool axis and the rotation axis of the workpiece are arranged perpendicularly, it is called orthogonal turn milling (Fig. 9.43, left). Alternately, in axis-parallel turn milling, both rotation axes are parallel to each other as shown in Fig. 9.43, right. Due to the arrangement of the milling tool, axis-parallel turn milling makes both internal and external machining possible.

In the case of orthogonal turn milling, the milling cutter executes a screw-shaped motion relative to the workpiece due to the axial feed movement and the rotation of the shaft. In accordance with this characteristic, DIN8589c defines turn milling as a variant of screw milling.

We also draw a distinction between centric and eccentric turn milling (Fig. 9.44). In centric turn milling, the workpiece axis and tool axis intersect at one point, while in the case of eccentric turn milling they are offset relative to each other by a certain amount, eccentricity *e*.

High surface quality (low facet formation) can be obtained in turn milling only with extreme milling cutter and shaft speed settings.

This can be realized with a centric cutter setting (e = 0) (Fig. 9.44):

- by lowering the feed per workpiece rotation – and thus the cutter engagement – to a few 1/100 mm. We work simultaneously with very rapid workpiece rotation and slow cutter rotation. In this way, we obtain a process very similar to turning.
- by very slow workpiece rotation and fast cutter rotation [Köni84a, Köni84b, Köni85, Köni86]. The workpiece speed can be lowered so far that the quickly rotating cutter edges successively cut towards the target radius almost continuously. For the sake of high cutting performance, the feed per workpiece rotation – and thus the engagement width of the milling cutter – should be maximized.

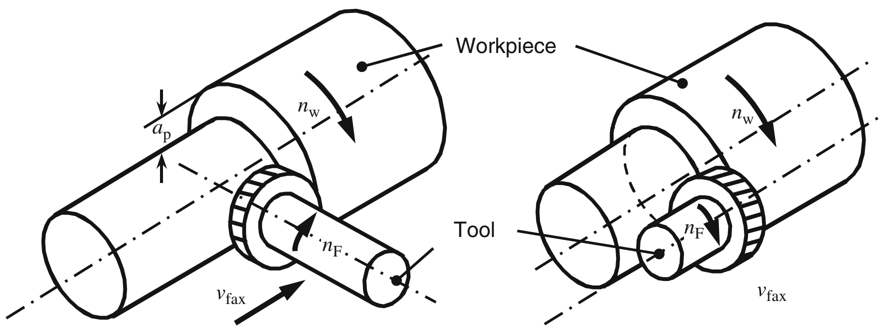


Fig. 9.43 Axle rotation in turn milling

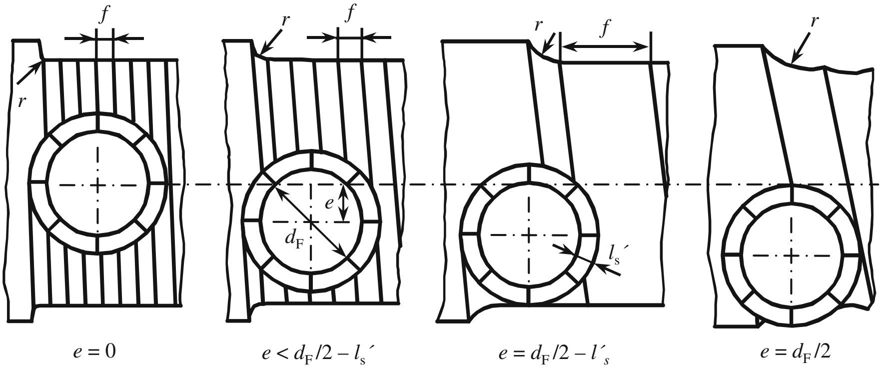


Fig. 9.44 Centric and eccentric turn milling

In order to produce cylindrical surfaces, a broad-tool finish tool geometry with a face cutting edge lead angle of $\kappa'_r = 0°$ is unavoidable, since from now on the face cutting edges create the workpiece target radius with their full length. The feed per workpiece rotation f_{ax} is thereby limited to the length of the face cutting edge L_s' [Köni84b]. Under these conditions, the process is comparable to conventional face milling.

In the second case, the axial feed can be increased beyond the limit set by the length of the face cutting edge by shifting the milling cutter by the eccentricity e without hazarding cylindricity faults. In the case of an eccentricity of approximately

$$e = \frac{d_F}{2} - L_s' \tag{9.6}$$

the maximum of feed of

$$f_{ax} = 2\sqrt{\left(\frac{D}{2}\right)^2 - e^2} \tag{9.7}$$

is reached. Further augmentation of the eccentricity again reduces the adjustable axial feed until finally no cylindrical surfaces can be produced [Köni84a, Köni84b]. A great advantage of centric turn milling is that sharp-edged workpiece shoulders can be created (Fig. 9.43). In eccentric turn milling, this is impossible for kinematic reasons. Shoulders are principally rounded in this case. In addition, the surface quality is not as good as in the centric process variant. Centric turn milling is thus of great interest for practical process design despite lower performance values. It is recommended therefore to implement roughing operations with an eccentrically and finishing operations with a centrically positioned milling cutter [Köni86].

Superimposition of the rotary motions of the workpiece and tool as well as of the feed motion results in highly complex cutting kinematics [Köni84a, Köni84b, Köni85]. Figure 9.45 shows an example of this process.

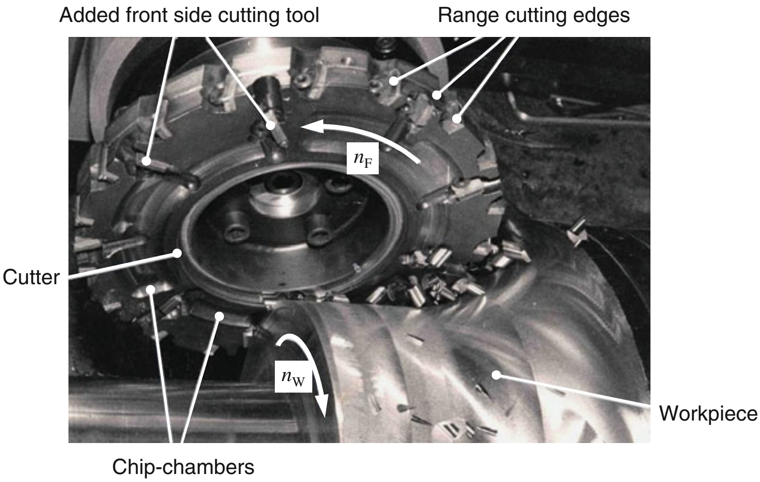


Fig. 9.45 The turn milling process (Source: Wohlenberg, Seco)

As opposed to conventional face milling, the face cutting edges participate actively in the machining process and each produce one element of the workpiece surface, as the material is moved into the front face of the milling cutter by means of the workpiece rotation. The facet-like surface typical of turn milling can clearly be seen (Fig. 9.45). In the case of the special tool used here, the face cutting edges are extended with additional inserts in order to maximize the feed. To avoid cylindricity faults, these also have a lead angle of $\kappa'_r = 0°$. An effective and economical cutting process is also possible by using conventional milling heads. In steel machining in a smooth longitudinal cut, TiN-coated cemented carbides have proven to be particularly effective. The cutting conditions are generally somewhat higher than those of conventional face milling with TiN-coated cemented carbides [Wand92]. Steel materials that are difficult to machine can on the other hand be very effectively machined with TiCN-coated cemented carbides by means of turn milling [Stal94].

The particular advantages of turn milling are safe chip fracture when machining long-chipping materials as well as the high cutting performance in a smooth longitudinal cut at low workpiece rotary speeds. This is especially significant in the case of large or unbalanced parts. Turn milling can be used to create circular profile curves (e.g. on camshafts) and cylinder forms that run eccentrically to the component rotary axis (e.g. lifting pins on crankshafts). Lifting pins and crankshaft cheeks can be machined with externally or internally geared tools (Fig. 9.46). A flexible full-range processing of such components in one clamping is possible by exploiting the additional rotary axis for drilling, slab milling or thread cutting [Köni84b, Kauf92].

Beyond that, various mould parts can be created (e.g. longitudinal grooves) by adjusting the shaft rotary motions with the three possible, mutually independent feed motions (axial: f_{ax}, radial: a_p, tangential: e) [Köni84a].

Fig. 9.46 Crankshaft machining with rotary milling (Source: Walter)

Extruder screws are almost exclusively produced by turn milling or planetary thread milling [Sten64]. Spherical actuators for pipeline valves as well as ellipse and eccentric forms can also be manufactured via turn milling.

9.3 Drilling

The term drilling signifies the machining method with a rotary main motion in which the tool is allowed only one feed motion in the direction of the tool rotary axis. The most important process variants are shown in Fig. 9.47 along with the common respective directions of motion [DIN8589b]. The peculiarities of drilling include:

- a cutting speed that falls to zero towards the drill centre,
- difficult chip removal,
- unfavourable heat distribution at the action point,
- increased wear at the sharp-edged cutting edges and
- friction of the lands against the drill hole wall.

Different objectives with respect to material removal rate, drill depth, dimensional accuracy and surface quality have led to the development of a series of different drilling processes, which will be explored in more detail in the following.

9.3.1 Profile Counterboring

Counterboring differs from drilling basically by the fact that one does not drill into solid material but rather a prepared hole which has, for example, been drilled or punched, is counterbored to a dimension smaller than specified or to finished dimensions.

DIN 8589-2 distinguishes between to process variants:

- planar countersinking
- planar insertion

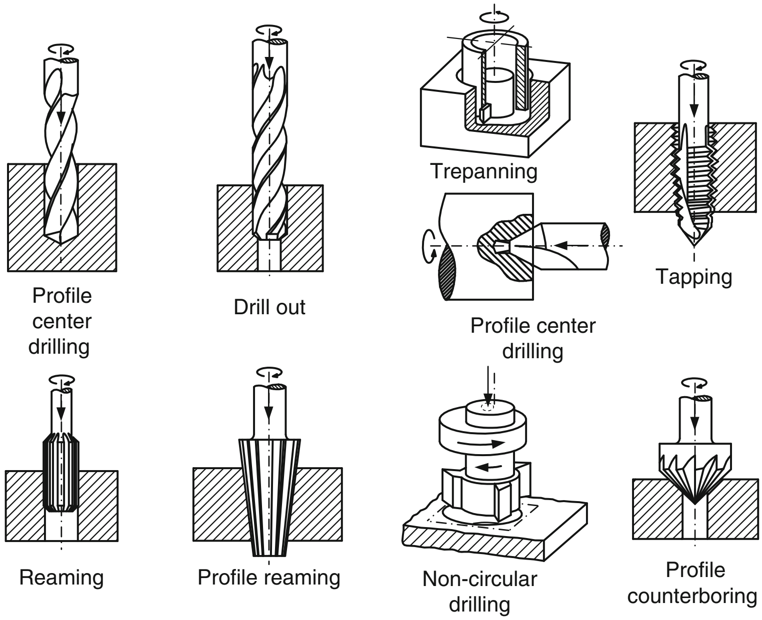


Fig. 9.47 Process variants of drilling, according to DIN 8589

In the case of planar countersinking, an even surface protruding on the workpiece lying perpendicular to the rotary axis of the cutting motion is created. Planar insertion on the other hand is used to produce an even surface recessed in the workpiece lying perpendicular to the rotary axis of the cutting motion, whereby an internal cylindrical surface is formed simultaneously. For planar insertion of pre-cast or predrilled drill holes, usually three-lip spiral countersinks are used. In comparison to spiral drills, the three-blade, screw threaded design gives the spiral countersink much higher stiffness and thus leads to much higher working precision.

For deburring, chamfering and inserting the seat of spherical screw heads, HSS countersinks are used that are manufactured according to standard with angles of taper σ of 60, 90 and 120° (Fig. 9.48).

For manufacturing drill holes of fastening screws, screw head counterbores of HSS or cemented carbide design are employed, the shape and size of which is adjusted to the respective standardized screw type.

Piloted counterbores are suitable not only for planar insertion but also for planar countersinking the front faces of eyes and hubs.

Automation in manufacturing often presumes the use of tools that are adjusted to a particular machining task. Such special tools, to which the form countersink in Fig. 9.48 belongs, can shorten production times quite considerably, since several working cycles can be consolidated in one spindle stroke. For example, it is customary in mass production for a screw connection to drill one through-hole with a form tool, then to counterbore the cylinder for the head of a hexagon socket screw and finally to chamfer the drill hole edge.

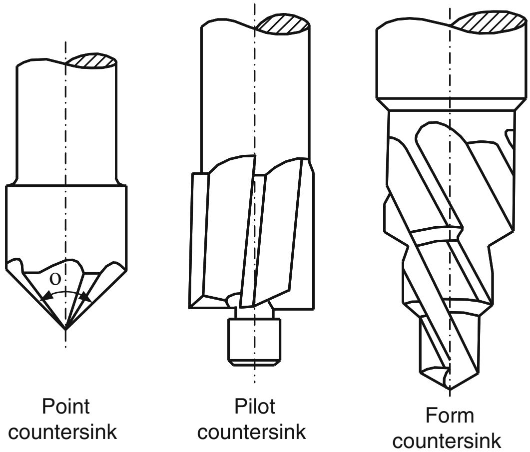


Fig. 9.48 Counterboring tools

9.3.2 *Rotary Drilling*

9.3.2.1 Centre Drilling

The spiral drill occupies the position of greatest importance among drilling tools, as it is the most important tool for creating cylindrical drill holes from solid material or for enlarging a preset drill hole diameter in drilling out. It is estimated that it takes up 20–25% of machining operations, and it is today the machining tool that is produced in the largest numbers and is the most widespread [Häus79, Tika93].

Simply put, the spiral drill is composed of the shaft and the cutting part (Fig. 9.49). Only a more exact consideration reveals the complex geometric formation, especially of the drill bit. The about 150 grind types [Tika93] and numerous material-specific drill profiles represent the attempt to do justice to multifarious machining task with respect to quality and performance. For some time, analytical models have been developed to help calculate tool stresses. This requires an

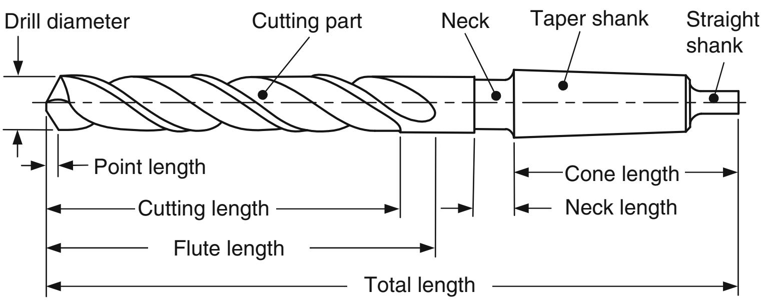


Fig. 9.49 Spiral drill with taper shank, acc. to DIN 1412

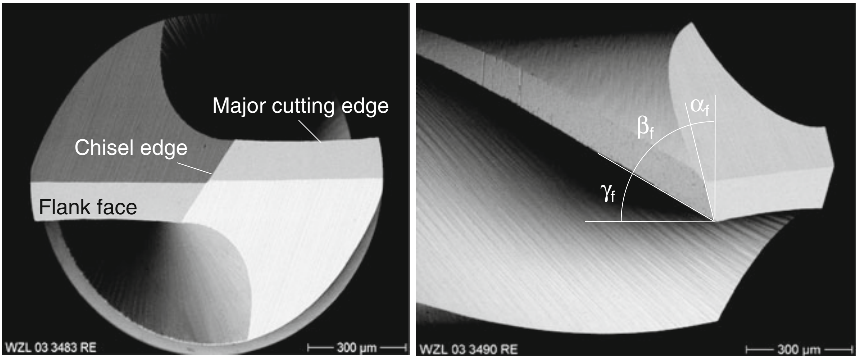


Fig. 9.50 Cutting part geometry of a spiral drill

acquaintance with the cutting part geometry, the kinematics of the drilling process and the stresses arising during that process.

Figure 9.50 shows the cutting part geometry of a spiral drill. Since according to the definition the major cutting edges point in the feed direction, the chisel edge is also part of the major cutting edge, although it hardly cuts due to its highly negative tool orthogonal rake angle, but rather deforms the material plastically and forces it to the major cutting edge.

The shape and pitch of the chip flutes determine the size of the tool orthogonal rake angle γ_o, which is not constant along the major cutting edge but decreases from its highest value on the corner (γ_f) towards the drill centre and becomes negative in the transition to the chisel edge (Fig. 9.51). The only differentiating factor however that is used is the side rake angle γ_f, which is identical to the helix angle δ with sufficient accuracy. The latter is varied because of the differing chip fracture of different materials and categorized into the main drill groups N for normal materials, H for hard materials and S for soft materials (Fig. 9.52) [DIN 1414a].

By means of the interaction of the cutting motion (rotation) and the feed motion, the tool cutting edge moves along a screw line. Taking the cutting conditions (effective speed) into consideration, the tool orthogonal clearance must be selected such that the effective rake angle is positive. An upper boundary of the clearance is also given however by the weakening of the cutting part and the rattling tendency.

To machine steel materials, usually a point angle of 118° is chosen. Point angles of 90° are used for drilling hard, usually heavily wearing plastics in order to make the transition from the major cutting edge to the lands less sharp than is the case with very large point angles, thus reducing edge dulling correspondingly. Point angles of 130° result in improved free drilling in the case of resilient ("clamping") materials; moreover, the chip clogging problem can be countered with a further enlargement to a point angle of 140° in the case of long-chipping light metals (Fig. 9.53).

In summary, it can be seen that only a careful, automatically executed drill point grinding adjusted to the particular problem can lead to an economical machining

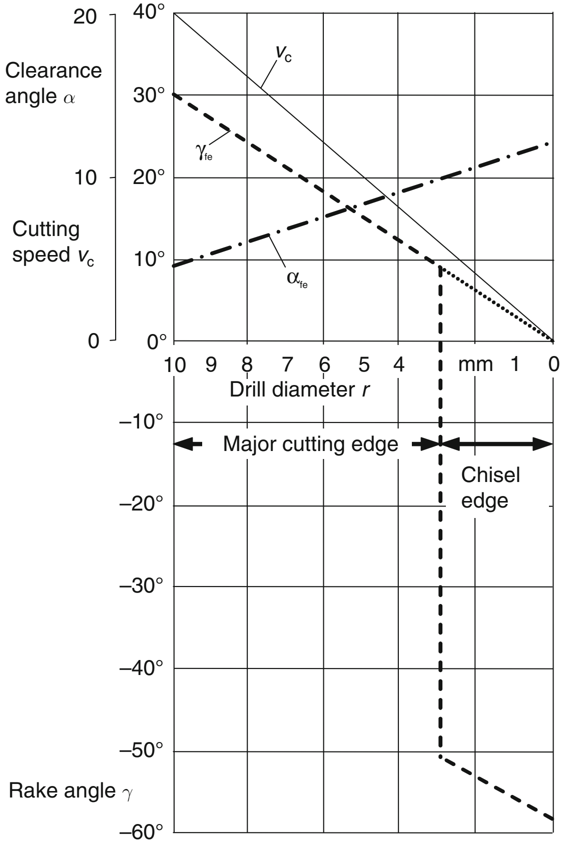


Fig. 9.51 Rake angle, clearance and the cutting speed against the drill diameter

process. For the larger part of all machining cases, the conical relief point has asserted itself as the most consistent and suitable type. The rake faces are parts of a taper sleeve (Fig. 9.54). The advantages of these drills include easy manufacture and preparation as well as their low sensitivity to high mechanical stress.

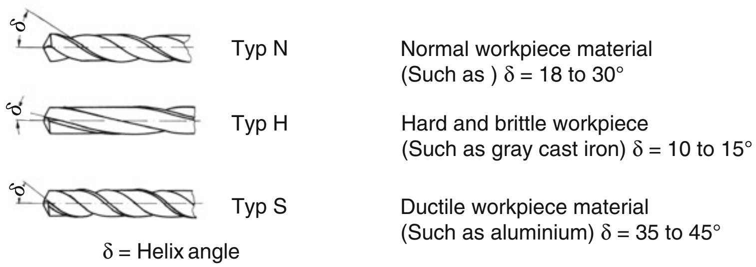


Fig. 9.52 Spiral drills for various materials

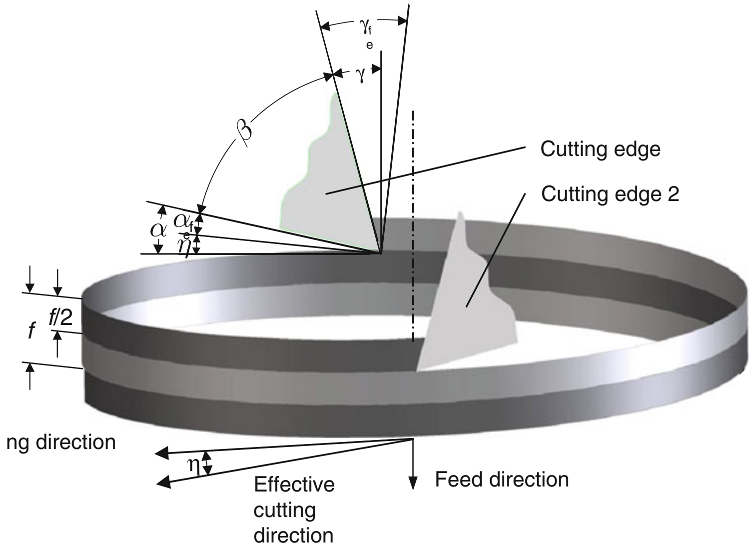


Fig. 9.53 Motion sequence of the major cutting edges by double edged drilling tool

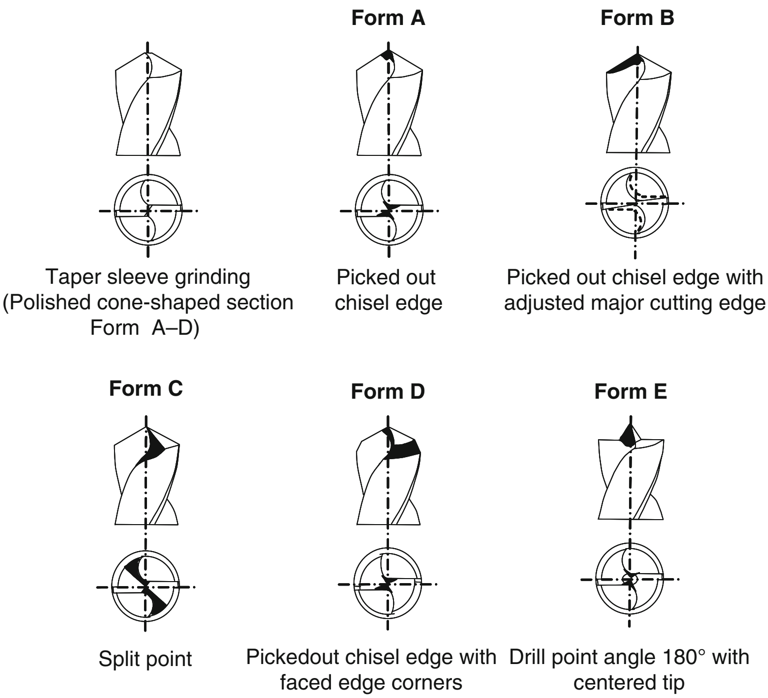


Fig. 9.54 Combination of spiral drills with special polished section from model A till E compared to taper sleeve grinding, acc. to DIN 1412

The low self-centring and associated shape and position errors are disadvantageous. In addition, the chisel edge length is increased with increasing drill and core diameters, such that the resulting high feed forces have an unfavourable effect on machining accuracy.

In this case and generally only when special demands are placed on the drilling tool, the drill point is equipped with a special grinding that either complements the taper sleeve grinding (e.g. core point thinning) or completely reshapes the drill point (centre point, Fig. 9.54).

The following describes the most important point grindings in accordance with Fig. 9.54:

- Form A: the taper sleeve grinding with a point-thinned core improves to a great extent the centrability of the drill and decreases the axial force corresponding to the chisel edge shortened by about $0.1{\cdot}D$ (used in general for Type N beyond 14 mm diameter).
- Form B: the taper sleeve grinding with point-thinned core and corrected rake angle makes it possible to adjust the rake angle to the machining task. However, it is customary to reduce the rake angle by about 10°, resulting in a very stable wedge without hindering chip transport because of a diminished helix angle. Grinding B is used in cases of high drill stress such as encountered when machining austenitic manganese steel or when drilling thin-walled aluminium sheets to reduce deformation.
- Form C: A taper sleeve grinding with a split point in which case the chisel is completely eliminated. This is especially suitable for deep drill holes. The compressive chisel edge is converted into two small major cutting edges with much better cutting properties. This type also guarantees good centrability and reduced feed force.
- Form D: The taper sleeve grinding with point-thinned core and bevelled corners was specially developed for machining grey-cast iron workpieces, the hard, abrasive casting skin of which stresses the sensitive corner to a particularly large extent. Here, a second taper sleeve grinding with a smaller drill-point angle provides a remedy in that it helps improve heat conduction and counters increased wear by increasing its surface area.
- Form E: drill-point angle 180° with centred tip, used when centric drilling must be guaranteed or when round and burr-free drill holes are to be made in aluminium sheets. After total penetration of the centring cone, both major cutting edges simultaneously cut to their full length, and the corners can support themselves immediately on the drill hole wall with the lands. The drill exits again by the entire major cutting edge, whereby a ring-shaped disc is cut out with minimal burr formation.

The four-face grinding, a taper sleeve grinding with a secondary face, is mentionable despite the fact that it is not standardized inasmuch as it is used when drilling below 1.5 mm diameters or with cemented carbide drills, since in this case the tapered sleeve grind causes difficulties.

The special working conditions of a drilling tool place high demands on the cutting tool material with respect to hardness, toughness, wear resistance and insensitivity against thermal alternate stresses. Frequently, HSS is used as a cutting tool material. According to DIN 1414-1, high speed steels for drilling tools, contain 6% tungsten, 5% molybdenum, 2% vanadium (HS6-5-2) and for higher stresses 5% cobalt (HS6-5-2-5). The tools are hardened, topically treated (nitrated) and often equipped with wear-preventing coatings.

Solid cemented carbide drills are also used however. The advantages of cemented carbides are their high hardness, compressive strength and high-temperature wear resistance. Cemented carbides have the same hardness at 1000 °C as high speed steel at room temperature. As a rule of thumb, the cutting speed v_c can only be increased by a factor of three. Beyond that, the machining process can continue with a feed f that is at least 30% higher. Besides this increase in cutting conditions, the tool life travel path L_f can be extended by a factor of three. Due to their high Young's modules, cemented carbide drills are much more torsion-stiff than HSS tools.

Due to their high hardness and low toughness compared to HSS tools, their use is technically meaningful and economical only on machine tools that fulfil the minimum requirements regarding accuracy, power, cooling and stiffness. One example for precision requirements is the concentricity of the drilling process. The total radial deviation measurable at the cutting edges of the drill is the result of the sum of each radial deviation of the machine spindle, interface, tool holder and tool. In current practice, the tool holder has the highest share. If the minimum requirements cannot be fulfilled, HSS drills are still preferred, not the least because of their lower price.

Drilling as a machining process has several peculiarities which we will examine in the following. In comparison to internal turning, drilling with spiral drills produces a greater surface finish on the drill hole wall, which is the result of the comparatively low cutting speed, the low torsion and bending stiffness of the tool and chip transport [Spur60]. Moreover, spiral drills are subject not only to flank face and crater wear but also chisel edge wear, land wear and corner wear.

In the case of the spiral drill, total tool wear is composed basically of

- flank face wear,
- crater wear,
- land wear,
- chisel edge wear and
- corner wear

and leads ultimately to relative or absolute disruption of the tool.

Relative disruption is characterized by the fact that beyond a certain drilling length the machining output no longer correspond to the requirements. In the case of absolute disruption, the HSS tool becomes completely unusable because of thermal induced failure of the tool cutting edge or because of tool fracture.

One essential tool life criterion in drilling is reaching pre-given limits in the case of dimension and shape faults (relative disruption). Wear on the corner and on the lands is frequently responsible for this. Due to the maximum cutting speed on the outer diameter, the corner is especially stressed. This is where HSS tool often fail. On the other hand, the low cutting speed in the chisel edge area often causes built-up edge formation, which however does not have a dominantly negative effect on the process sequence nor makes itself perceptible in the output.

The feed influences tool wear much less than the cutting speed, so it will no longer be considered here. The forces acting on the spiral drill are represented in Fig. 9.55. When drilling with HSS tools, the cutting speed for steels is in the range of 10–40 m/min. A series of studies has shown that here the influence on the resultant force is small, especially in the case of large drill diameters. In the case of extremely low or extremely high cutting speeds on the other hand, a considerable increase in the feed force and cutting moment has been noted, which has an especially large effect when smaller diameter drills are used.

As in all machining processes with geometrically defined cutting edges, the resultant force components increase degressively with the feed (Fig. 9.55).

Exhaustive investigations have shown that the surface quality of the drill hole wall cannot be significantly affected by the drill point grinding. On the other hand, the dimensional accuracy of the drill hole is dependent on the symmetry of the grinding, since straying of the tool can only be prevented when there is an extensive balancing of the radially acting passive force [Spur60].

Different passive forces F_{p1} and F_{p2} subject the drill to bending and lead to enlarged drill hole diameters in the output. Such passive forces arise primarily for tool-related reasons, such as

- unequal major cutting edge lengths,
- unequal drill-point angles,
- unequal tool orthogonal clearances,
- asymmetrical point thinning,

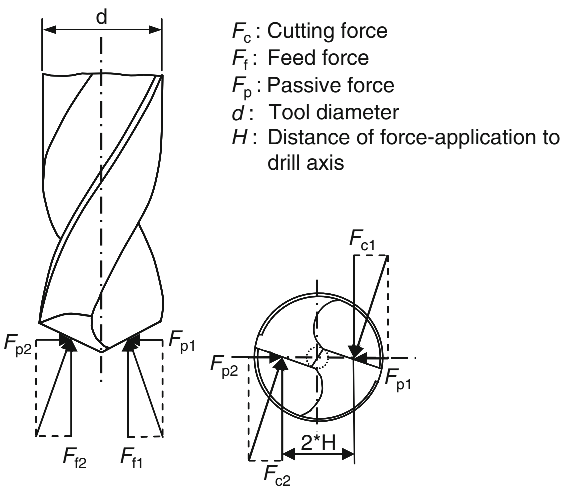


Fig. 9.55 Spiral drill forces, according to [Spur60]

Table 9.1 Force components of the major cutting edge, the chisel edge and the land of a spiral drill

	Torque (%)	Feed force (%)
Major cutting edge	65–75	17–25
Chisel edge	10–14	65–75
Land	15–20	7–8

- asymmetrical spiral flutes,
- inconsistent cutting edge sharpness and
- radial deviations [Spur60].

The increased cutting and feed forces compared to turning (at otherwise identical marginal conditions) are due on the one hand to friction of the chips in the bottom of the groove and on the drill hole wall and on the other hand to the length of the chisel edge, which should be referred to in consideration of the resultant force more than the drill diameter. The more narrow the chip space is for chip volume specified by the cutting conditions the larger are the required cutting moment and the necessary feed force. Among the feed forces that arise during turning, considerable amounts of force are also added due to friction on the land and compressive processes in the area of the chisel edge.

The guidelines in Table 9.1 should be referred to when considering the percentage proportion of force of the major cutting edge, the chisel edge and the land of a spiral drill with respect to torque and feed force:

These percentages were determined by means of step drill experiments. The torques and feed forces can be determined sequentially only for the major cutting edges, for the major and chisel edges as well as for the major edge, chisel edge and land.

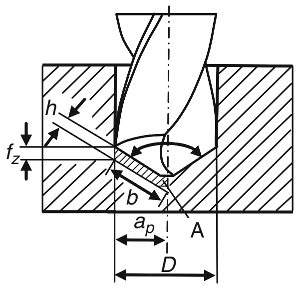
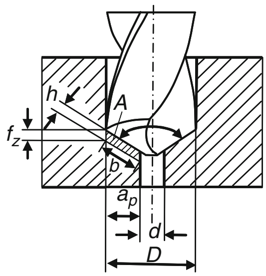
9.3.2.2 Cutting Parameters in Drilling

The cross-section of undeformed chip A has a major influence on the resultant force in drilling. Table 9.2 shows the relevant relations of the cutting parameters in drilling and the potential calculation of the cross-section of undeformed chip from the feed component per cutting edge f_z and the depth of cut a_p or from the chip thickness h and chip width b.

9.3.2.3 Calculating Forces, Torque and Power When Drilling with Spiral Drills

Analogously to the ratios in drilling, we can approximately calculate the cutting force, feed force and torque for centre drilling and drilling out with spiral drills with the help of the KIENZLE equation (Table 9.3). For drilling, it is necessary to introduce a process factor f_B in order to take into consideration the altered influences on the forces that occur in drilling as opposed to turning (e.g. cutting edge shape, cutting speed etc).

Table 9.2 Calculation of the cross-section of undeformed chip in drilling

	Centre drilling	Drilling out
Cutting parameters		
	$f_z = \frac{f}{z}, \kappa_r = \frac{\sigma}{2}, b = \frac{a_p}{\sin(\kappa_r)}, h = f_z \cdot \sin(\kappa_r)$	
	$A = f_z \cdot a_p = b \cdot h$	
Cross-section of undeformed chip	$A = \frac{d \cdot f}{4}$	$A = \frac{(D-d) \cdot f_z}{2}$

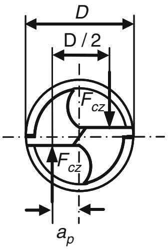
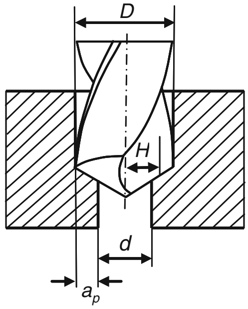
D	Drill diameter [mm]	f	Feed [mm]
d	Pre-hole diameter [mm]	f_z	Feed per cutting edge [mm]
a_p	Depth of cut [mm]	κ_r	Lead angle [°]
b	Chip width [mm]	σ	Drill-point angle [°]
h	Chip thickness [mm]		
z	Number of cutting edges		

9.3.2.4 Deephole Drilling

Deephole drilling is a machining process used to produce or process drill holes. Deepholes are drill holes with a diameter between about 1 and 1500 mm and a drilling depth of about three times the diameter. It is not possible to make a general distinction between deephole drilling and other "conventional" drilling techniques by means of a universally valid definition or the like. With all deephole drilling methods, a very large ratio of drilling depth to diameter can be obtained. Further advantages of deephole drilling compared with customary drilling with spiral drills include above all the higher quality of the drill holes and its excellent cost-efficiency.

Not only is deephole drilling distinguished from common drilling by its asymmetrical cutting edge arrangement, but also by the fact that in deephole drilling a cutting fluid is fed directly to the cutting edges under pressure and that its rinsing effect is the sole transport mechanism for the incoming chips. The cutting part is made of cemented carbide, so high cutting speeds can be reached, which in turn

Table 9.3 Forces, torque, power required in drilling

	Centre drilling	Drilling out
Force application	$H = D/4$ $a_p = D/4$  	$H = (D + d)/4$ $a_p = (D - d)/4$ 
Process factor f_B	$f_B = 1$	$f_B = 0.95$
Cutting force per cutting edge F_{cz}	$F_{cz} = \frac{(D - d)}{2} \cdot f_z \cdot k_c \cdot f_B$	$F_{cz} = \frac{D}{2} \cdot f_z \cdot k_c \cdot f_B$
Feed force per cutting edge F_{fz}	$F_{fz} = \frac{D}{2} \cdot f_z \cdot k_f \cdot f_B$	$F_{fz} = \frac{(D - d)}{2} \cdot f_z \cdot k_f \cdot f_B$
Torque	$M_d = \frac{F_{cz} \cdot z \cdot D}{4000}$ For $z = 2$: $M_d = \frac{F_{cz} \cdot D}{2000}$ $M_d = \frac{9554 \cdot P_c}{n}$	$M_d = \frac{F_{cz} \cdot z \cdot (D + d)}{4000}$ For $z = 2$: $M_d = \frac{F_{cz} \cdot z \cdot (D + d)}{2000}$
Power	$P_a = \frac{P_c}{\eta}$ $P_c = \frac{F_{cz} \cdot v_c}{60000}$	$P_c = \frac{M_d \cdot n}{9554}$ $P_c = \frac{F_{cz} \cdot v_c (1 + d/D)}{60000}$

f_B	Process factor drilling	k_f	Specific feed force [N/mm^2]
H	Lever [mm]	F_{cz}	Cutting force per cutting edge[N]
D	External drill diameter [mm]	M_d	Torque [Nm]
d	Internal drill diameter [mm]	P_a	Drive power [kW]
z	Number of cutting edges	P_c	Cutting power [kW]
f_z	Feed per cutting edge [mm]	n	Speed [min^{-1}]
k_c	Specific cutting force [N/mm^2]	v_c	Cutting speed [m/min]
		η	Efficiency

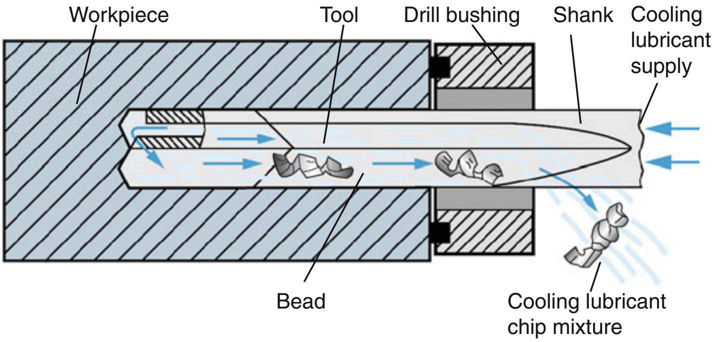


Fig. 9.56 The ELB process for diameters 0.8–40 mm, according to Sandvik

makes it possible to increase the material removal rate. The following three process variants are used for the industrial production of deep drill holes:

- the single-lip drilling process (ELB process),
- the BTA drilling process,
- the ejector drilling process.

The single-lip drilling process is used in the diameter range of approximately 0.8–40 mm. Figure 9.56 shows the essential characteristics of this method.

The characteristic trait and main advantage of the single-lip drilling process is that the cutting fluid is supplied by means of one or several drill holes within the tool and the cutting fluid/chip mixture is safely led away by a longitudinal groove (bead) on the outside of the tool shank.

Due to the shape of the lead and the large drilling depth/diameter ratio, the drill is guided on the top face of the workpiece by means of a drill bushing (Fig. 9.56). This stabilizes the start of drilling. Single-lip deephole drilling tools are used in manufacturing as solid drills, core drills, countersinks and step drills, whereby full drilling is the most common case of operation in practice. Basically, the single-lip drill consists of three components: the drill head, the shank and the clamping sleeve. In most cases, cemented carbide is used as the cutting tool material, whereby both solid cemented carbide drill heads as well as drill heads fitted with cemented carbide are employed.

The BTS process (Boring and Trepanning Association – BTA) was invented at the end of the 1930s in order to prevent chip scratching on the drill hole wall during transport and the resulting damage to the surface quality. The attempt to cover the flute of the single-lip drill outwards resulted however in a drastic reduction of available chip space, which in turn limited the material removal rate. The solution was finally discovered by the "Boring and Trepanning Association", who reversed the process characteristics of single-lip drilling and supplied the cutting fluid from outside by means of a ring-shaped crack between the drill pipe and the wall (Fig. 9.57). Reflow occurs together with the chips through the cutting jaw and the drill pipe, the

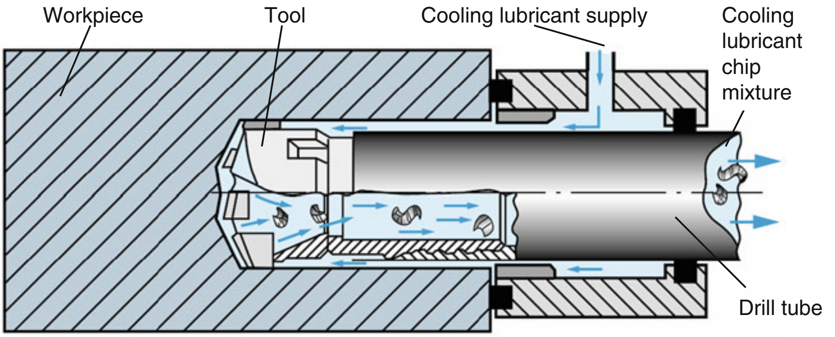


Fig. 9.57 The BTA process for diameters of 6–300 mm, according to Sandvik

diameter of which should not be less than 6 mm. The upper diameter for full drilling tools is around 300 mm and for countersink tools around 1000 mm, whereby these limits depend to a large extent on the available machine power.

Compared to deephole drilling with single-lip drills, the BTA process has the disadvantage that a complicated drill oil supply apparatus is required which takes over the sealing of the drill pipe. The process requires machines that are much different than standard drill machines.

The ejector deephole drilling process is utilized for diameters of about 18–250 mm. According to VDI guideline 3209, it is a variant of the BTA process (Fig. 9.58). Cutting fluid supply is accomplished by means of a ring space between the drill pipe and an internal pipe (two-pipe process). The cutting fluid enters the drill head from the side, rinses it and flows back into the internal pipe with the chips. Part of the cutting fluid is introduced into the internal pipe via a ring nozzle. Reflow is made possible by the arising low pressure at the cutting jaw (ejector

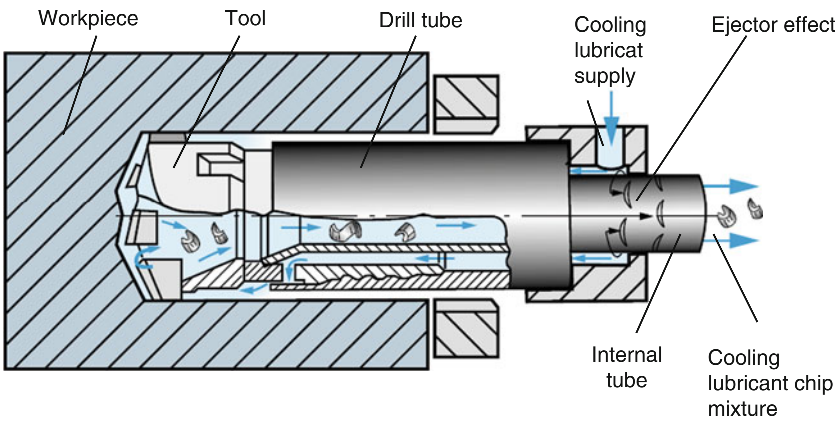


Fig. 9.58 The ejector process for diameters of 18–250 mm, according to Sandvik

effect). As opposed to the BTA process, sealing against the exit of the cutting fluid is omitted. A further peculiarity is its cutting edge distribution for reducing the forces acting on the guide beads as well as the double-sized cutting jaw thus required. The cutting edge, otherwise continuous from the periphery to the centre, is subdivided such that two cutting parts are arranged at a time alternating left and right up to the centre. The consequence of this is that the stress on the guide beads is reduced by about 10% to a maximum of 50% of the otherwise expected forces, and friction, heat development and wear are reduced accordingly.

Deephole drilling tools dominate the entire field of inner contours that can be manufactured by drilling (Fig. 9.59). Other drilling methods and tools are used only

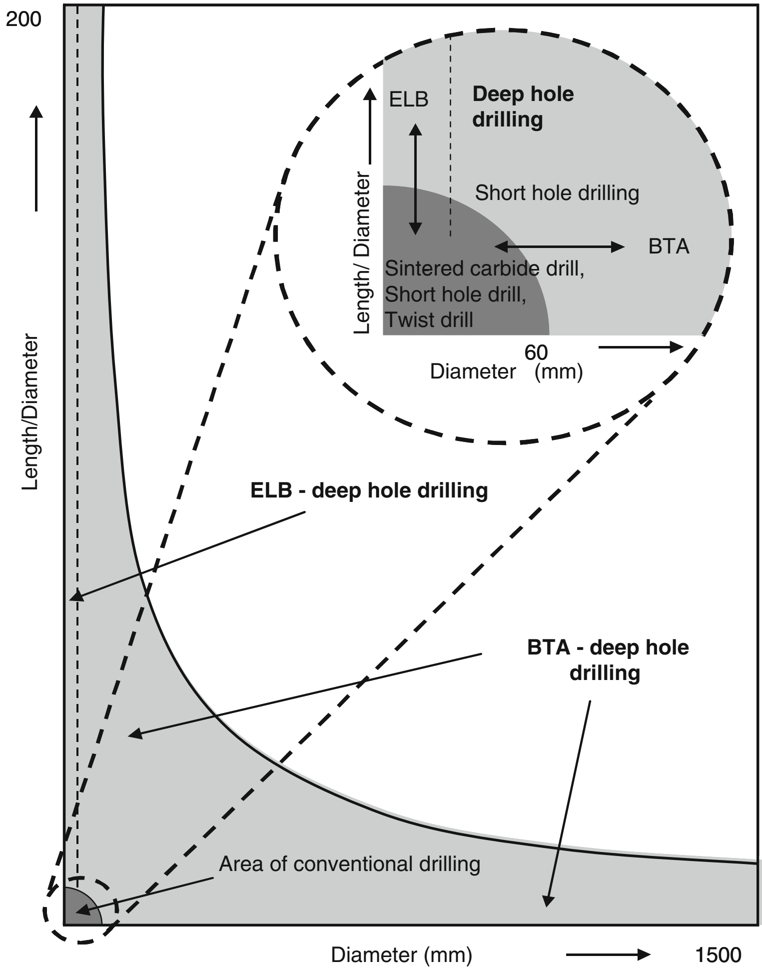


Fig. 9.59 Application of the deep hole tools compared to "conventional" drilling tools, acc. to VDI guideline 3210

in the range of smaller drilling depths (up to a length/diameter of ca. 6 and a diameter of up to ca. 60 mm). Since these dimensions are predominant in general mechanical engineering, the dominance and versatility of deephole drilling processes is often not perceived. Increasingly, especially in the area of overlap between short-hole drilling (conventional drill technology) and deephole drilling, tools are being used that have the characteristics of deephole tools or are operated under conditions that resemble a deephole drilling operation.

Especially in the field of deep drill holes and drill holes with large diameters, deephole techniques are used now almost exclusively. Due to its high productivity and the drill hole quality obtainable, deephole drilling is being used increasingly for manufacturing tasks in which the ratio between the drill hole depth and the drill hole diameter is larger than 6. Numerous examples of machining reveal the presence of deephole technology in the area of smaller tool diameters as well, in which naturally most application cases for drilling are to be found. Difficult-to-machine materials can as a rule be machined effectively with deephole methods.

Deephole processes have the following typical characteristics:

- the use of special cemented carbide tools with one or sometimes more cutting edges that lie asymmetrically to the tool axis,
- self-commutation of the tool by means of a three-point mounting in the drill hole through guide beads and the minor cutting edge (cylindrical grinding chamfer),
- drill start guidance of the tool in a drill bushing or a guiding bore,
- continuous high-quantity cutting fluid supply under pressure resulting in constant chip removal without chip removal strokes.

Some advantages of deephole drilling are:

- very high machining performance,
- ideal cooling and lubricating conditions,
- short primary processing times,
- high drill hole quality with respect to diameter tolerance, surface quality and geometric contouring accuracy,
- high alignment accuracy, minimal drill hole inaccuracy,
- replacement of several operations – e.g. pre-drilling, boring and reaming – by one single operation,
- possibility of processing hard-to-machine materials,
- large drilling depths in relation to the diameter (up to a maximum of 250 times larger),
- cost-efficiency, even with short drilling depths,
- minimal burr-formation when drilling out and when overdrilling cross-holes.

By means of deephole drilling, metals of all kinds as well as other materials (e.g. plastics) can be processed both in the mass production of small parts and in the single-part production of large-scale machine parts.

The process variants (including machining methods) of deephole drilling are characterized by the drilling task and the correspondingly adjusted drilling tools

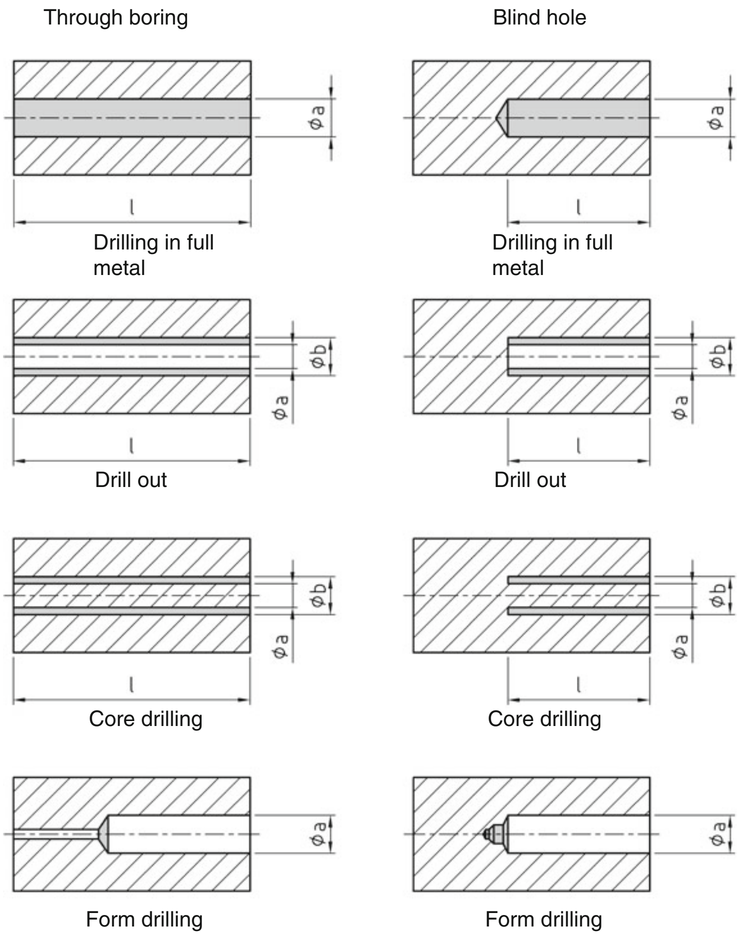


Fig. 9.60 Deephole drilling process variants

(Fig. 9.60, VDI guideline 3210). The most frequently utilized variant of drilling is full drilling. Moreover, all deephole drilling processes can be employed for drilling out and core drilling. Form drilling tools serve to produce a definite drill hole bottom defined by the tool cutting edge geometry or contour transitions in the case of step drilling.

Deephole drilling machines can be constructed in a way that the primary motion can be carried out by either the tool or the workpiece or by both. In the case of a rotating tool and a stationary tool, the machines are applicable for a broad range of arbitrarily shaped parts. With an automatic loader and potentially as multi-spindle machines, they are most suitable for economical large-batch production. For both rotating and stationary tools, only rotation-symmetrical parts with small masses can be used, since there is a danger that even a small unbalance of the rotating

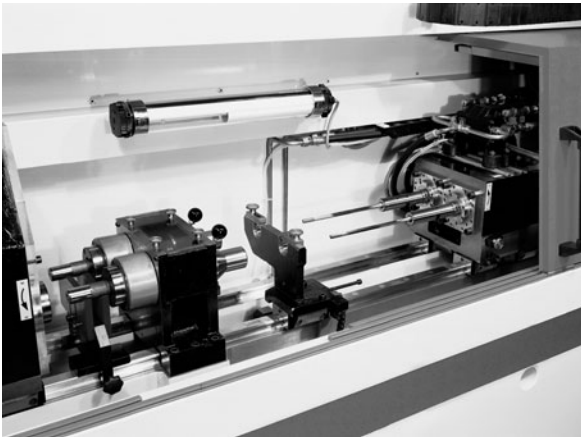


Fig. 9.61 Deephole drilling machine (Source: TBT Tiefbohrtechnik)

workpieces may lead to poor drilling results. Figure 9.61 shows a deephole drilling machine by the company TBT Tiefbohrtechnik. Some of the technical data of this deephole drilling machine are:

- full drilling rage (min.–max.) : 0.9–15 mm,
- maximum drill depth with 1 steady rest : 700 mm,
- maximum spindle speed : 24.000 1/min,
- drive power : 2.4 kW.

The limits of use of deephole drilling processes are essentially determined by the following factors:

- the machinability of the material,
- the stability of the tool and the machine,
- the accuracy of the machine,
- the composition of the cutting fluid,
- the cutting tool material.

VDI guideline 3210 summarizes these limits in the form of standard values for full drills, core drills and countersinks and distinguishes according to the process and tool whether ISO degrees of tolerance above or below IT9 can be reached. This approximate indication is grounded in the fact that the machinability of the material is still clearly part of the obtainable tolerance (Fig. 9.62).

For example, non-ferrous metals permit an ISO tolerance of IT6 under optimal conditions, which corresponds to a diameter range of 50–80 mm of a maximum deviation of 9 μm. On the other hand, the best possible tolerance when machining nitriding steels is IT8.

Since the process combination spiral drilling/reaming can be substituted with deephole drilling, under normal conditions the surface qualities obtainable are in

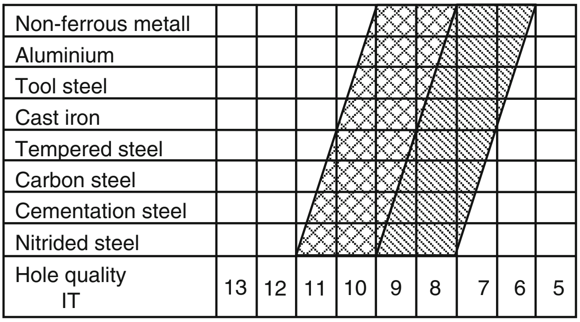


Fig. 9.62 Tolerances obtainable in deephole drilling (Source: TBT Tiefbohrtechnik)

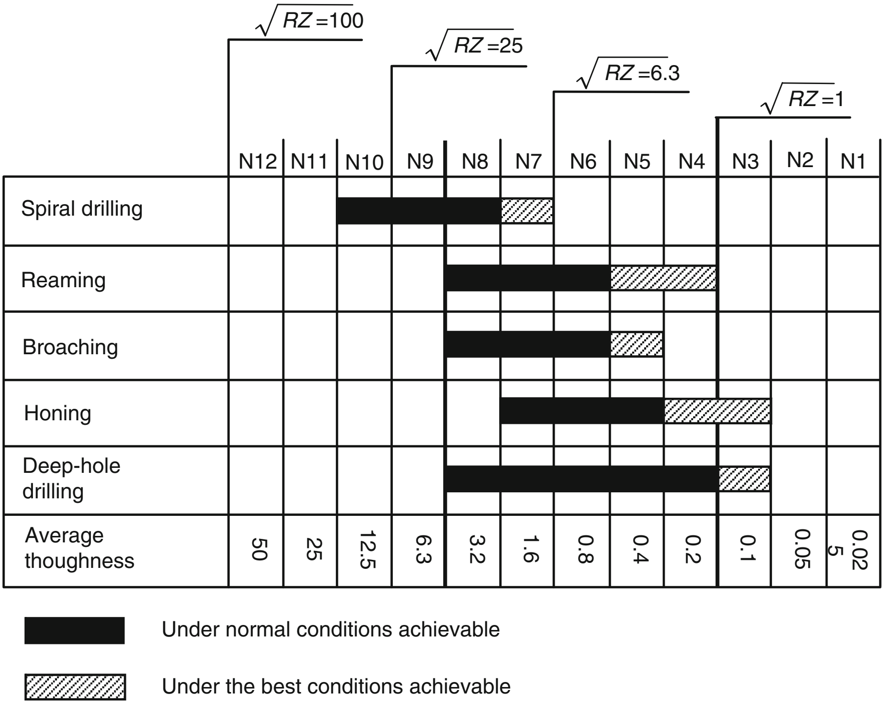


Fig. 9.63 Obtainable surface quality in comparison to other machining methods (Source: TBT Tiefbohrtechnik)

the same order of magnitude (fine finishing). Figure 9.63 shows furthermore that under especially favourable conditions even R_a values of 0.1 μm are obtainable, which otherwise require a superfinishing operation such as honing.

We can see from these considerations that deephole drilling is especially economical when subsequent processes can be reduced and/or high-alloyed materials are to be processed.

The following overview summarizes the areas of application in which deephole drilling can be used advantageously [Grüb74]:

- high material removal rate requirements
- machining materials with high alloy components that are hard to machine
- materials with a tensile strength of over 1200 N/mm^2
- high tolerance and surface quality requirements
- large drilling depth in relation to the diameter
- substitution of several single process steps (full drilling, drilling out, reaming) with one process step.

9.3.2.5 Reaming

Reaming is a fine finishing process and serves to improve drill hole quality, whereby position and shape errors cannot be influenced. With respect to kinematics, reaming is equivalent to drilling out with small chip thicknesses (Fig. 9.64).

According to DIN 8589-2, a distinction is drawn between reaming with single-blade and multi-blade tools. The single-blade reamer is guided by a guiding bead arranged on the periphery, whereby the functions of machining and guiding are divided among independent active elements (Fig. 9.65). The multi-blade reamer is guided by the minor cutting edge arranged on the periphery.

Fig. 9.64 Reaming tools, according to [DIN 8589-2]

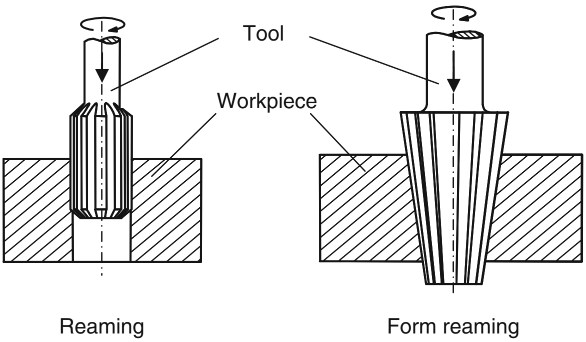


Fig. 9.65 Drill out tools with PKD – equipping and also with ISO – indexable inserts, according to Mapal

The cutting edges of multi-blade reamers can be arranged parallel to the axis or on a helical line. Drill holes with grooves are reamed with spiral tools in order to avoid cutting engagement impact of the cutting edge.

Usually, reamers are manufactured with an even number of teeth, whereby two cutting edges face each other at a time, which makes it much easier to determine the diameter. In order to prevent clattering vibrations, an odd distribution of cutting edge distances is selected, which repeats after half of the circumference. Drill hole qualities of IT7 or better are obtainable.

A distinction is made between manual reaming and machine reaming. In the case of manual reamers, the cutting tool material used is usually tool steel or HSS, while machine reamers use high speed steels or cemented carbides. Performance can be enhanced by employing coated tools in reaming as well. Insert changeability makes it possible to adjust the tool to different materials and machining tasks by means of an appropriate choice of substrate, coating and geometry.

9.3.2.6 Internal Thread Production

Internal threads can be manufactured by means of primary shaping, forming or cutting. Besides the available technology, essential considerations when selecting the manufacturing method include the material to be machined, the thread type and the number of units required as well as the required tolerances, strengths and surface quality.

Among the process variants used, cutting manufacture via tapping takes the leading position because of it is the most widespread.

9.3.2.7 Tapping

Tapping is drilling out for the manufacture of an internal thread that lies coaxially to the rotation axis of the cutting motion (Fig. 9.66).

Screw taps consist of a shank and a screwed portion. In the case of the screwed portion, a distinction should be drawn between the cutting part where machining occurs and the guide part, which is responsible for stabilizing the tool [Zura90].

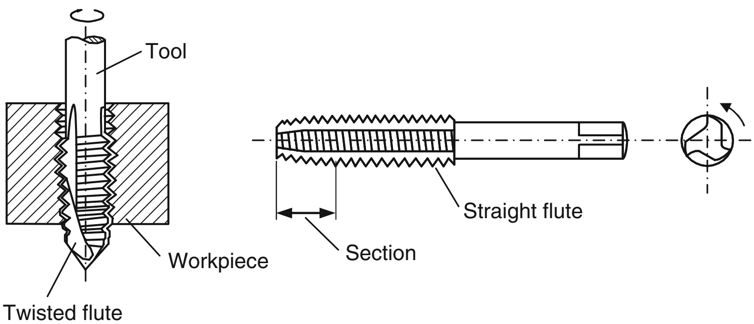


Fig. 9.66 Screw tap

Screw taps for through borings have a lead that guarantees good tool guidance. The screwed portion is subdivided into cutting studs by grooves. The grooves serve to receive the chips and convey them outside. Moreover, they guarantee cutting fluid supply to the cutting location. Large groove cross-sections facilitate this and are especially important when machining long-chipping materials. However, they lead to a weakening of the load-bearing tool cross-section.

To process short-chipping materials, screw taps with even grooves, which are easier to manufacture, are sufficient. In the case of long-chipping materials, spiral chip spaces are necessary to facilitate chip removal and to reduce the danger of chip jams. For the sake of good tool centring, at least three cutting studs are required, and thus three chip flutes as well.

In the case of manual screw taps, a set of two or three tools (pre-cutters and finish cutters or pre-cutters, intermediate cutters and finish cutters) are used to produce the thread in order to minimize tool load and the risk of fracture.

The cutting tool materials employed in tapping are high speed steels and cemented carbides. The surfaces of the screw taps can be processed to increase their wear resistance. The most important processes used are nitriding, hard chrome plating and coating with hard materials. The potential cutting speeds are relatively low and depend considerably on the combination of cutting tool material, workpiece material and cutting fluid.

9.3.2.8 Thread Milling

Under certain conditions, thread milling can be a good alternative to other thread manufacturing processes. Thread milling is a special screw milling process. It can be used, for example, to obtain very good surfaces on the screw flanks and to produce large numbers of units economically. Threads with large diameters can often only be fabricated via milling (number of units, tools costs, power input) [Fosh94].

The structure of a screw milling cutter is basically similar to that of a "screw tap". As opposed to the screw tap, which consists as it were of a single spiral-shaped tooth, the consecutive teeth of a screw milling cutter do not form spirals but rather are arranged without offset (Fig. 9.67).

The tooth shape corresponds as a rule to the form of the thread to be formed. In many cases it is necessary to correct the tooth profile. This is the case when the thread to be milled is not at least three times larger than the milling cutter's diameter. Without a correction of the profile, the tool would cut freely, distorting the finished thread profile. A screw milling cutter can mill threads of various diameters. It is not

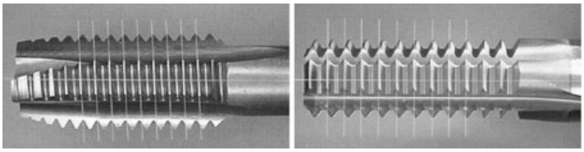


Fig. 9.67 Screw tap with coil (*left*) and milling cutter without coil (*right*), according to Fraisa

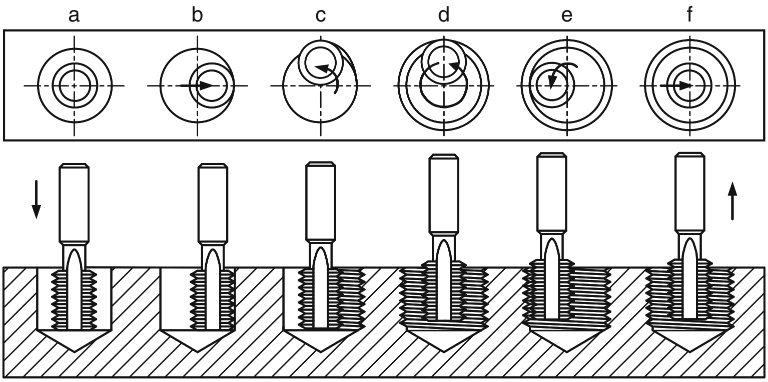


Fig. 9.68 The thread milling cycle

possible however to vary the pitch. The screw milling cutter is thus designated by the standard thread diameter that corresponds to this pitch.

Figure 9.68 provides a graphical depiction of a milling cycle. The rotating milling cutter positioned in the centre of the drill hole is axially lowered to the desired thread depth into the core drill hole (Fig. 9.68a).

The tool is then adjusted to the drill hole diameter (Fig. 9.68b). The screw milling cutter is finally radially advanced to the required standard diameter of the thread (Fig. 9.68c), so that there is a defined axis distance between the tool and the drill hole axis. To form the thread, there is a movement cycle of somewhat more than 360° on a coil (Fig. 9.68d), whereby the workpiece or the tool is axially shifted around a pitch. Finally, the screw milling cutter is backed out of the thread radially via a circular arc (Fig. 9.68e), driven back to the cutter axis and lifted axially out of the thread (Fig. 9.68f).

9.3.2.9 Thread Moulding

According to DIN 8583-5, thread moulding is a non-cutting (forming) process in which the internal thread is created by impressing a tool (the thread moulder or groover) into the workpiece [Fieb95]. The thread moulder is separated axially into three sections. It has a conical lead part on the top which extends across several thread turns. The largest amount of forming work takes place on this area. The next section of largest external diameter has the function of removing the thread flanks from the mould. The following, slightly tapered calibration section serves to smooth the fabricated thread flanks and to guide the tool. The profile of a thread moulder is, in contrast to the round profile of a screw tap, a polygon with three or more flattened corner areas. The material is displaced on these forming edges, which serve to receive the lubricant. The thread moulding process does not require chip flutes, resulting in increased bending strength due to the larger cross-sectional area.

The starting situation in the case of thread moulding is also a pre-drilled hole with a diameter corresponding approximately to the flank diameter of the thread. While

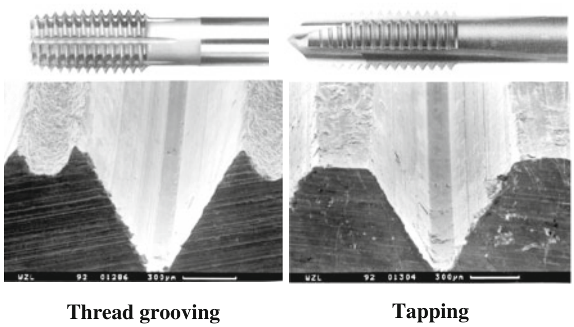


Fig. 9.69 Thread tools and several formations of flanks

the forming edges of the tool penetrate into the workpiece material and the thread flanks form to the required dimensions, the displaced material flows into the tooth gaps of the thread moulder. This causes the formation of ears, typical of formed threads, in the area of the thread crests (Fig. 9.69 left).

In comparison to “tapping”, thread moulding results in threads of higher strength. The cause is to be found in the strain-hardening that occurs during forming.

9.4 Sawing

Sawing is cutting with a rotary or translatory main movement with a multi-blade tool of low cutting width, used for separating or slitting workpieces. Sawing is classified as a process with a rotary main movement, since even in the process variants hacksawing and bandsawing, in which there is a translatory cutting motion, the saw blades can be seen as a tool with an infinitely large diameter (Fig. 9.70).

The following will be organized according to the type of tool used into bandsawing, hacksawing and circular sawing. These are the most commonly encountered process variants in praxis.

9.4.1 Bandsawing

Bandsawing involves a revolving, unending saw band cutting with a continuous, mostly linear cutting motion [DIN8589g].

The saw band is supplied on large rolls and, after the cutting to the desired length, the ends are joined together with a butt-welding process.

This process is distinguished by low cutting losses due to the small width of the bands. However this results as well in low tool stability against cut deviation.

Figure 9.71 shows the cutting part geometry and terminology of a saw tooth. The number of teeth is commonly given in reference to 1 in. of band length (ZpZ). The size of the chip space depends on the dimensioning of the tooth base radius *r* and on the number of teeth. As a result, when sawing larger material cross-sections, the tooth number has to be reduced in order to obtain a sufficient chip space for the arising chips.

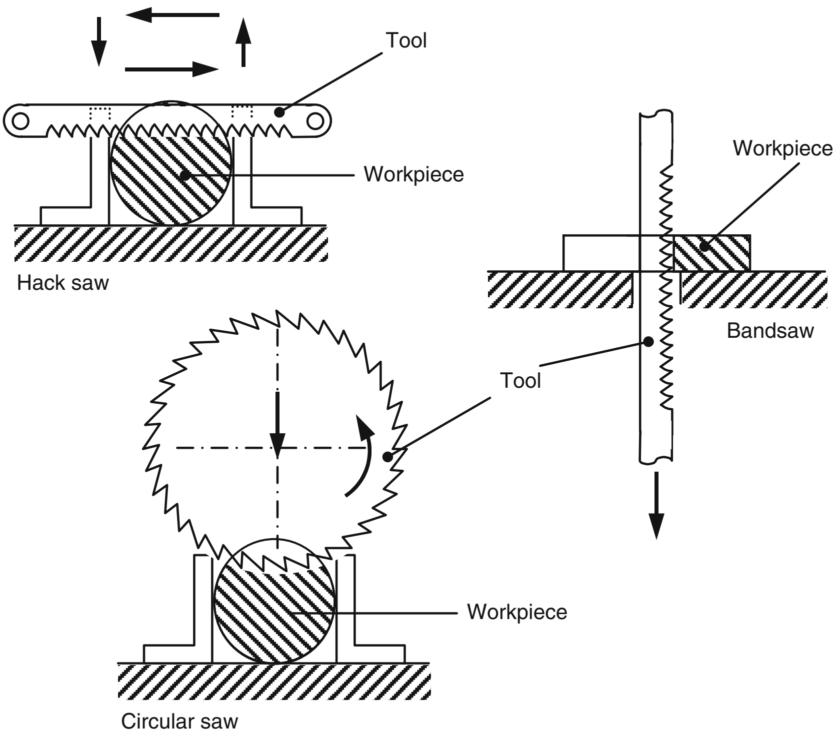


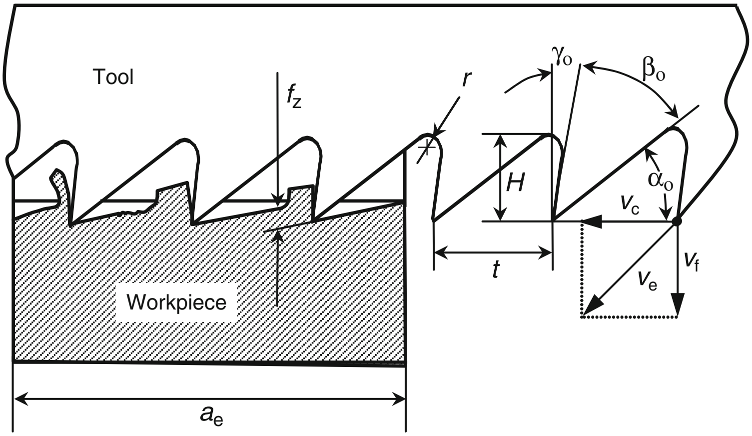
Fig. 9.70 Different sawing methods

In order to prevent jamming of the saw band in the cutting channel, the single teeth are bent or set to the left and to the right alternately from the cross section (see Chap. 3). The standard offset sequence for separating metals is right/left/straight. In the case of larger tooth numbers, shaft offset is also used (Fig. 9.72).

The cutting motion is parallel and the feed motion perpendicular to the longitudinal axis of the band. In the case of unlimited teeth, the depth of cut a_p corresponds to the width of undeformed chip b and in this case to the width of the saw blade. The engagement size a_e, measured as the size of the engagement of one cutting edge in the cross section perpendicular to the feed direction, corresponds to the workpiece width.

Saw bands are made either of tool steel or bimetal. In the case of bimetal bands, the blunt edge made of soft tool steel is joined by means of electron beam welding with a hardened band made of HSS, into which finally the saw teeth are inserted. Possible HSS cutting tool materials include HS6-5-2, HS2-10-1-8 or HS10-4-3-10. In practice, bands with soldered cemented carbide plates have also become established.

Tool life criteria that define the end of a sawband's usage are a certain wear condition, tooth fracture or a maximum permissible deviation of cut due to excessive



f_z	Tooth feed	v_c	Cutting speed
H	Tooth height	v_f	Feed rate
r	Base radius	v_e	Active speed
t	Tooth pitch		

Fig. 9.71 Title and part geometry on the cutting saw band, after RENG [Reng76]

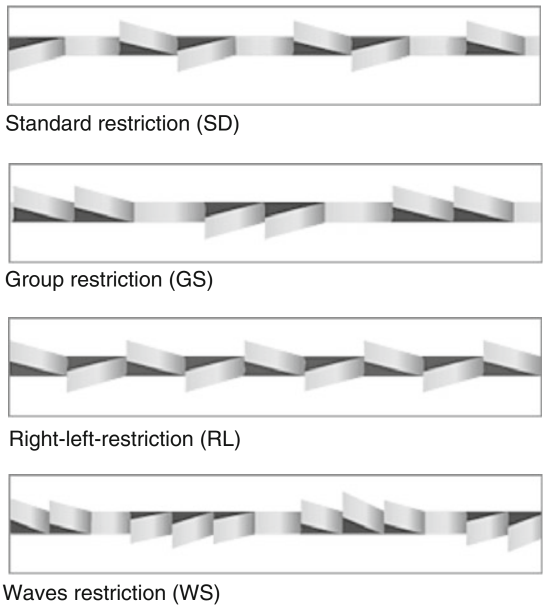


Fig. 9.72 Sawband restriction types (Source: Wikus)

passive force. In contrast to other cutting methods, the tool life is not given as the measure for the tool life parameter of a saw, but the cut surface (corresponding to the workpiece surface cut until reaching the end of tool life) [Reng76].

9.4.2 Hacksawing

Hacksawing is a process variant with a repeated, usually linear cutting motion (Fig. 9.70). According to DIN 8589-6, hacksawing, gangsawing, and jigsawing are all hacksawing processes [DIN8589f]. From this definition, we can already see that hacksawing is a process with a discontinuous cutting motion, i.e. with machine hacksaws, material is removed only in the forward stroke. In the return stroke, the saw blade is mechanically or hydraulically lifted. The result is that the material removal rate is lower compared with bandsawing or circular sawing. On the other hand, the cutting loss is relative small.

The saw blades are made either of solid HSS or of HSS segments that are riveted on a blade body. Blades made of tool steel are generally only used for manual hacksaws.

9.4.3 Circular Sawing

Circular sawing is sawing with a continuous cutting motion using a circular rotating saw blade (Fig. 9.73). This high-performance process is employed for linear cuts on low-cost materials, as much material is lost due to the relatively wide kerf.

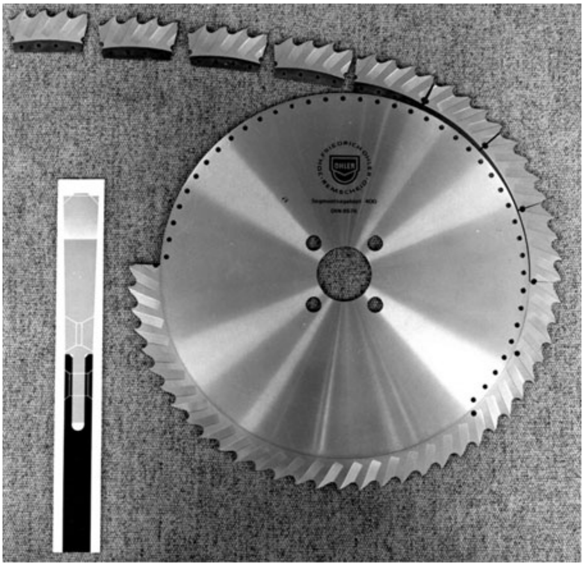


Fig. 9.73 Tooth segments of a saw blade (Source: Ohler)

Both tool steel and HSS as well as cemented carbides as used as cutting tool materials. At the same time, there are fundamental differences among construction types of saw blades. Saw blades made of one material are made of tool steel or HSS. In the case of larger saw blades. For cost reasons, the blade body is made of construction steel upon which the individual HSS segments are riveted (Fig. 9.73). The material removal can be improved further by means of soldered cemented carbide cutting edges.

In order to guarantee the cutting capacity of circular saw blades, it is absolutely necessary to break the chips in such a way that they are narrower than the kerf. If such measures are neglected, the chips jam in the chip space and can damage the tool. Figure 9.74 shows the two most prevalent possibilities [Schm80].

By subdividing the teeth into pre-cutting and post-cutting elements, the chips are fractured such that the pre-cutting teeth protrude at least by the amount of chip thickness and their major cutting edge length is smaller than the total width of undeformed chip.

One alternative is grinding-in offset chip breaker flutes into the otherwise identically formed teeth. In this way, one narrow and one wide chip is produced per tooth that can escape towards the centre into the chip breaker flute without jamming in the kerf. The fact that a tooth in the section of the chip breaker flute of the previous

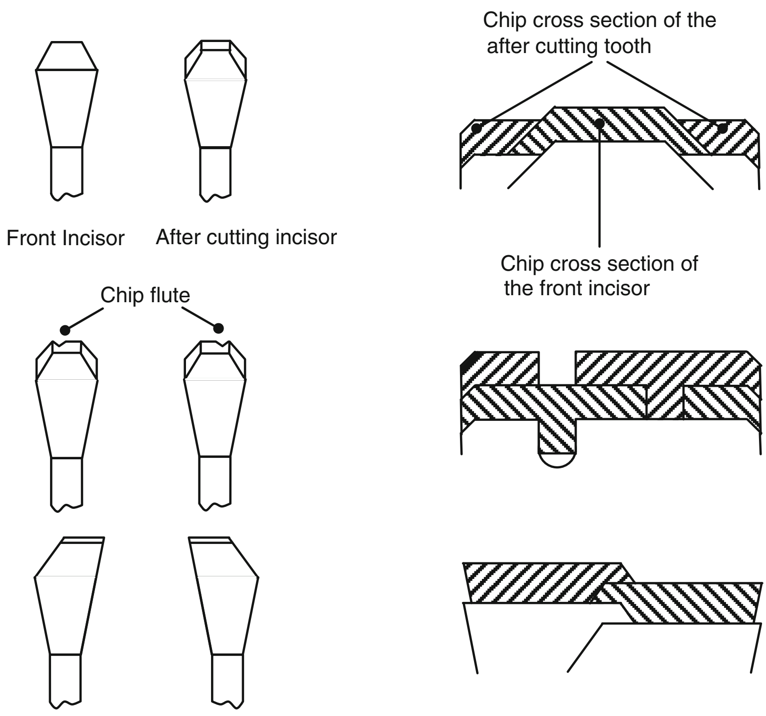


Fig. 9.74 Cutting edge geometries of circular saw blades

tooth must remove a larger chip cross-section has only an insignificant effect on the progression of wear. The advantage is that one tooth with a chip breaker flute can take on the same chip cross-section as the pre-cutting and post-cutting teeth of the other lead version.

In order to increase the circular saw blades' stability and running smoothness, it is necessary to introduce characteristic stresses into the blade in a specific fashion. This is done either by hammering or by rolling a concentric pressure zone into the side surfaces. A laser radiation process is also possible.

For separating general construction steel with HSS saw blades, values of $v_c =$ 18–30 m/min and $f_z = 0.22$–0.28 mm and with cemented carbide saw blades values of $v_c = 90 - 150\,\mathrm{m/min}$ und $f_z = 0.12 - 0.18\,\mathrm{mm}$ are common as cutting data.