Milling Applications And Cutter Basics

ISCAR's Reference Guide

Member IMC Group

www.iscar.com

CERTIFIED QMS AS 9100 THE STANDARDS INSTITUTION OF ISRAEL

CERTIFIED QMS SI ISO 9001:2015 THE STANDARDS INSTITUTION OF ISRAEL

CERTIFIED EMS SI ISO 14001:2015 THE STANDARDS INSTITUTION OF ISRAEL

CERTIFIED OHS SYSTEM SI ISO 45001:2018 THE STANDARDS INSTITUTION OF ISRAEL

CERTIFIED ENERGY MANAGEMENT SYSTEM SI ISO 50001 THE STANDARDS INSTITUTION OF ISRAEL

Quality Standard

ISCAR has been certified by the prestigious Standards Institution, as being in full compliance to ensure delivery of the finest quality goods. Quality control facilities include the metallurgical laboratory, raw metal testing, an online testing procedure and a machining center for tool performance testing and final product inspection. Only the finest products are packaged for entry into ISCAR's inventory.

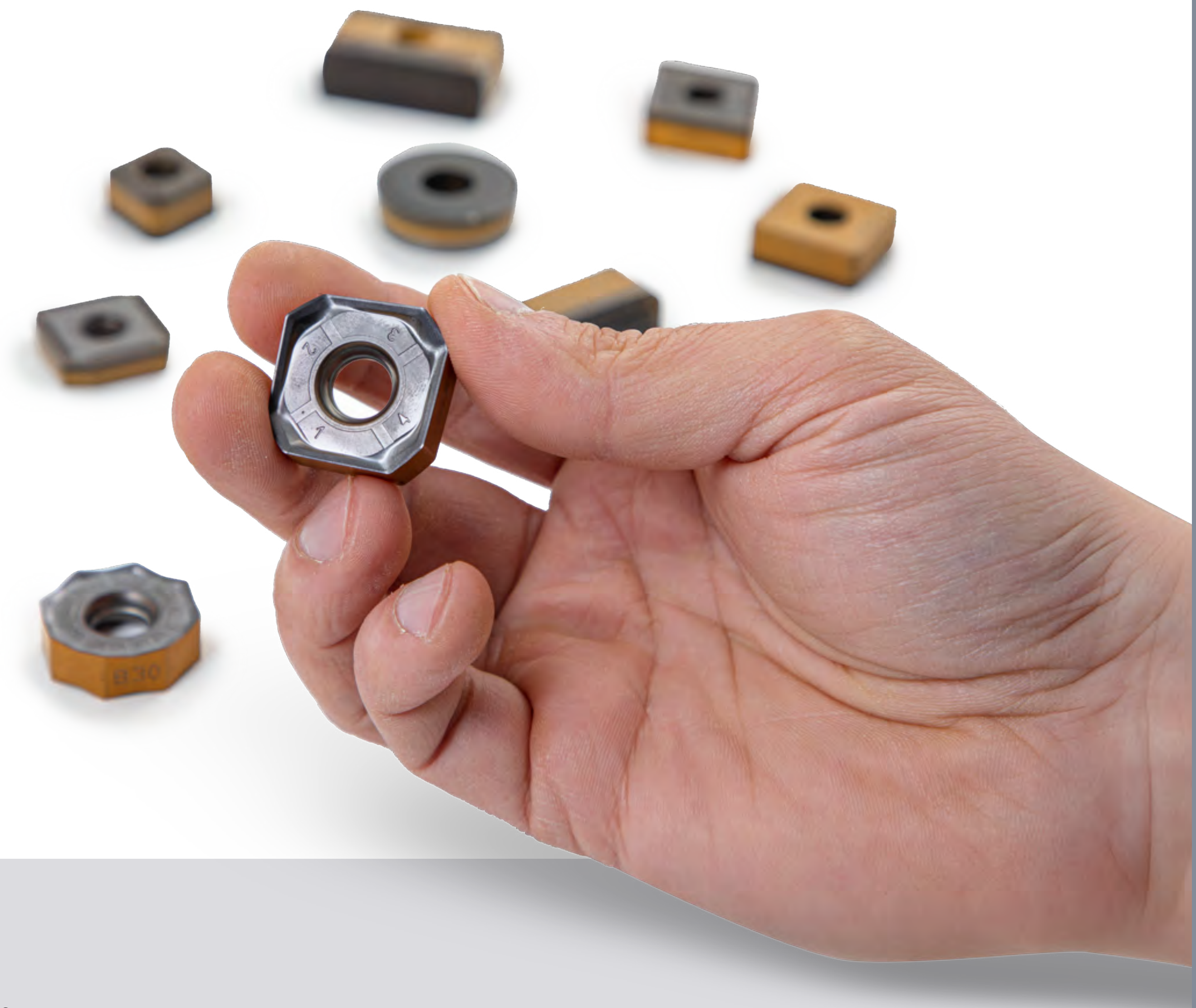
Member IMC Group

www.iscar.com

Contents

"Knowledge is a tool, and like all tools, its impact is in the hands of the user."

(Dan Brown)



Preface

This book serves as a continuation of the introduction to the world of cutting tools, initially presented in **ISCAR**'s reference guide, "Get to Know Cutting Tools." Now, the focus shifts to milling and milling tools, which are examined in detail. The aim of this book is to enhance understanding of motions, methods, techniques, cutter types, and many other aspects of milling. Designed as a pocket reference for various points related to milling, the book is also intended as a learning guide or tutorial. It is particularly useful for those who choose to improve their skills through **ISCAR**'s e-learning courses and aspire to become experts in the metal cutting industry.

The book includes self-evaluation quizzes for the main sections, which will assist you in assessing your progress in the self-study process.

Symbols and units*

- **A** – removed cross-section area, mm^2 (in^2)
- **AE** – angle of engagement
- **a** – depth of removed cross-section, mm (in)
- **a_e** – width of cut (radial depth of cut, w.o.c., WOC), mm (in)
- **a_p** – depth of cut (axial depth of cut, d.o.c., DOC), mm (in)
- **b** – width of removed cross-section, mm (in)
- **d** – cutting diameter, mm (in)
- **F** – total (resultant) cutting force, N (lbf)
- **F_a** – axial cutting force, N (lbf)
- **F_b** – bending force, N (lbf)
- **F_r** – radial cutting force, N (lbf)
- **F_t** – tangential cutting force, N (lbf)
- **f, f_r** – feed per revolution (feed), mm/rev (inch per revolution, ipr, IPR)
- **f_z** – feed per tooth, mm/tooth (inch per tooth, ipt, IPT)
- **h** – chip thickness, mm (in)
- **h_{max}** – maximum chip thickness, mm (in)
- **k_c** – actual specific cutting force, N/mm^2 (ksi)
- **k_{c1}** – specific cutting force to remove a material chip area of 1 mm^2 (.0016 in^2) with 1 mm (.004 in) thickness, N/mm^2 (ksi)
- **M_c** – cutting torque, Nm (lbf×in)
- **mc** – material factor in the equation for finding actual chip thickness
- **n** – rotational velocity (spindle speed), rpm, RPM
- **Q** – metal removal rate (MRR), cm^3/min (in^3/min)
- **P_c** – cutting power, kW (hp, HP)
- **v_c** – cutting speed, m/min (sfm, SFM)
- **v_f** – feed speed (feed rate, table feed, minute feed), mm/min (inch per minute, ipm, IPM)
- – clearance (clearance angle, relief, relief angle)
- **γ** – rake (rake angle)
- **γ_f, γ_r** – side rake (radial rake, side rake angle, radial rake angle)
- **γ_o** – orthogonal rake (orthogonal rake angle, effective rake)
- **γ_p, γ_a** – back rake (axial rake, back rake angle, axial rake angle)
- **η** – machine tool efficiency
- **κ** – cutting edge angle (entering angle, entrance angle)
- **κ'** – minor cutting edge angle
- **λ** – cutting edge inclination
- **λ_s, ω** – flute helix angle
- **φ** – angular pitch (pitch angle)
- **ψ** – lead angle (approach angle)

* in the metric system (units in the US customary system are given in brackets)

Abbreviation list

3D – three-dimensional
AM – additive manufacturing
CAE – computer aided engineering
CAM – computer aided manufacturing
CCW – counterclockwise
CBN – cubic boron nitride
CNC – computer numerical control
CPU – cost per unit
CW – clockwise
d.o.c. – depth of cut
FF – fast feed
ISO – International Organization for Standardization
HEM – high-efficiency machining (dynamic milling)
HFM – high feed milling
HRC – hardness, Rockwell scale C
HPC – high pressure coolant
HSM – high speed machining, high speed milling
HSS – high speed steel
HTSA – high-temperature superalloys
LH – left-hand
MQL – minimum quantity lubrication
MRR – metal removal rate
PCD – polycrystalline diamond
R&D – research and development
RH – right-hand
SCEM – solid carbide endmills
S/F – side and face
w.o.c. – width of cut

Milling and Milling Cutters Defined

Milling is a metal-removal process that involves using a rotating cutting tool to shape a workpiece, which moves in a translational motion relative to the tool.

A tool used for milling is commonly referred to as a “milling cutter”, “milling tool”, or simply “mill”. Most milling cutters are multi-point tools, meaning they have more than one cutting edge. The cutting edges of a mill are designated as the “teeth”.

Milling is a widely used metal-removal process that allows for the machining of workpieces in different sizes and shapes. In the production of parts that are not rotary bodies, milling is a common operation, used early in the manufacturing process to create the necessary datums for subsequent machining operations. In many cases today, finish milling can provide a surface finish that is comparable to that achieved through grinding.

While milling was initially developed for planar surfaces, advancements in multi-axis CNC machines now enable the creation of complex 3D surfaces through milling operations.

Motions in Milling

In milling, the primary motion is the rotary motion of a milling cutter, while the feed motion is the translational motion of a machined workpiece relative to the cutter. Depending on the type of milling machine, there are several ways to facilitate feed motion.

Table Feed

Traditionally, feed motion in milling was achieved by moving a machine table that held a machined workpiece. Consequently, the speed of the feed motion, known as the feed speed, was closely tied to the velocity of the moving table, commonly referred to as "table feed."

The most common approach involves moving the workpiece while the rotating milling cutter remains in the same position. Another technique is observed when the workpiece is stationary, and the rotating cutter moves translationally. Lastly, in the third approach, both the rotating milling cutter and workpiece are moved relative to each other.

Fig. 1. Example of motions in milling: primary motion is the rotation of a milling cutter with rotational velocity n, while feed motion involves the rectilinear movement of a machine table carrying a workpiece mounted on it, with a linear speed v_f.

Milling Methods and Milling Operations

The cutting edges of a milling cutter are located on the cutter periphery, cutter face, or both. Therefore, depending on the area of the cutter involved in the cutting action, there are several milling methods: peripheral milling, face milling, or a combination of both.

In peripheral milling, only the teeth on the outer periphery of the mill are engaged in cutting (Fig. 2). Peripheral milling is commonly known as 'slab milling', and the corresponding mill is referred to as a "cylindrical mill", "slab mill", or even "plain mill".

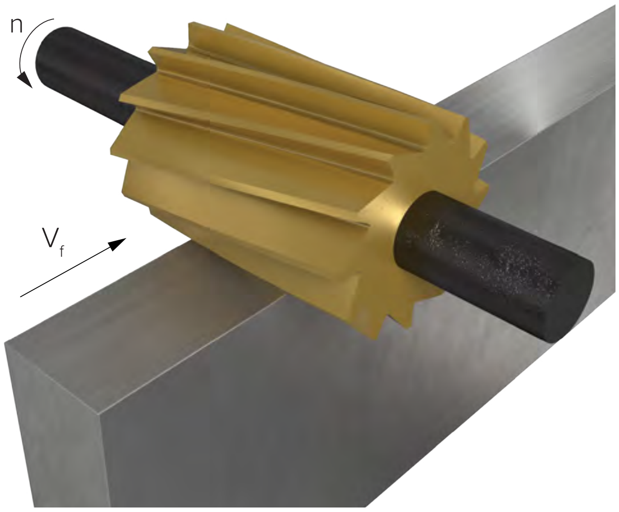


Fig. 2. Peripheral (slab) milling.

Slab Mill

A slab mill is a type of a cylindrical (plain) milling cutter – a milling tool with helical cutting teeth on its cylindrical periphery. slab mills generally feature large sizes and have a central bore for arbor mounting, mainly in horizontal milling machine tools. Slab mill length is considerably greater than its diameter. These mills are intended for machining an open surface (mostly plane) of a workpiece when the surface width is less than the mill length. Slab mills were very common in the past but today they are used quite rarely.

In face milling (Fig. 3), the cutting edges of a mill that perform the cutting action are located on both the mill end face and the mill periphery. A typical face mill, used for face milling operations, is a multi-toothed tool with a flat bottom. In face milling, the tool axis is aligned perpendicular to the machined surface, and the milling width is significantly larger than the milling depth.

Fig. 3. Face milling.

End milling is a milling technique that can be seen as a subset of face milling. Unlike face milling, end milling involves a greater milling depth compared to the milling width. If the applied endmills have a cylindrical shape, their tool axis runs parallel to the main machined surface, such as a shoulder, slot, or cavity wall. These endmills are sometimes referred to as face mills with extended tooth length. Fig. 1 provides an example of end milling a deep shoulder, while Fig. 4 illustrates end milling a shouldered contour.

Fig. 4. Milling shouldered surface by solid carbide endmill.

Profile milling is a general name for milling techniques used for machining three-dimensional (3D) surfaces. It is also known as “form milling,” “milling contoured surfaces,” or simply “profiling.” Special-shape milling tools, such as ball-nose, toroidal, and segment-shaped, are specially designed for profile milling (Fig. 5).
The development of multi-axis CNC machines has expanded the possibilities and potential for profile milling.

Slot or Groove?

The words “slot” and “groove” are often synonymous. But if “slot” usually relates to a narrow, comparatively long, mainly longitudinal, opening that is usually open-ended (at least from one side); “groove”, as a rule, means a circular (called “undercut”) or helical channel. It has been said that “a slot is an open-ended groove.”

Milling slots and grooves or simply slot milling can be accomplished using different tools, such as endmills and face mills. However, a disc-shaped side milling cutter, designed with teeth on its face and periphery, is specifically intended for this type of milling. This type of cutter allows for the simultaneous machining of three surfaces: the bottom and the two sidewalls of a slot (Fig. 6).

Consequently, these cutters are also referred to as “side-and-face (S/F) mills.” As a result, the milling technique is commonly known as “side and face (S/F) milling.

Fig. 5. Machining the complex surfaces of a propeller with a barrel-shape endmill.

Fig. 6. Milling slot by side-and-face milling cutter.

Plunge milling, also referred to as “plunge-in milling” or “plunging”, involves the feed motion of a milling cutter along the cutter axis. The cutter is then plunged into the material of the machined workpiece (Fig. 7).

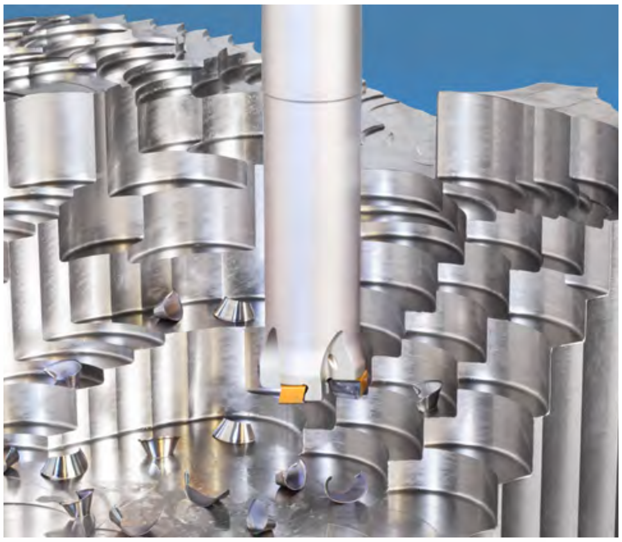


Fig. 7. Machining a workpiece by plunge milling

In addition, milling is used for cutting threads, gears, splines, and sprockets. However, these processes are typically treated as separate classes of technology (threading, gearing etc.).

Fig. 8 shows the main milling operations, and Fig. 9 illustrates them schematically.

Gang Milling and Straddle Milling

In gang milling, a combined tool consisting of two or more milling cutters, mounted on the same arbor, is used to machine multiple surfaces of a workpiece simultaneously. Straddle milling, a variation of gang milling, employs two or more side-and-face milling cutters, mounted on a single arbor, to machine parallel planes of a workpiece. These planes are perpendicular to the arbor axis and feature an exact distance between them. To ensure the required accuracy of this distance, the milling cutters are spaced apart with the use of bushings and spacers.

1 — Milling 3D Surface (Profile Milling)
2 — Milling Flat Face with Square Shoulder
3 — Face Milling
4 — Milling Slot by Face-and Side (Disc) Cutter
5 — Milling Slot by Face Mill
6 — Milling Square Shoulder
7 — Milling Slot by Solid Endmill
8 — Milling Deep Shoulder
9 — Plunge Milling
10 — Thread Milling

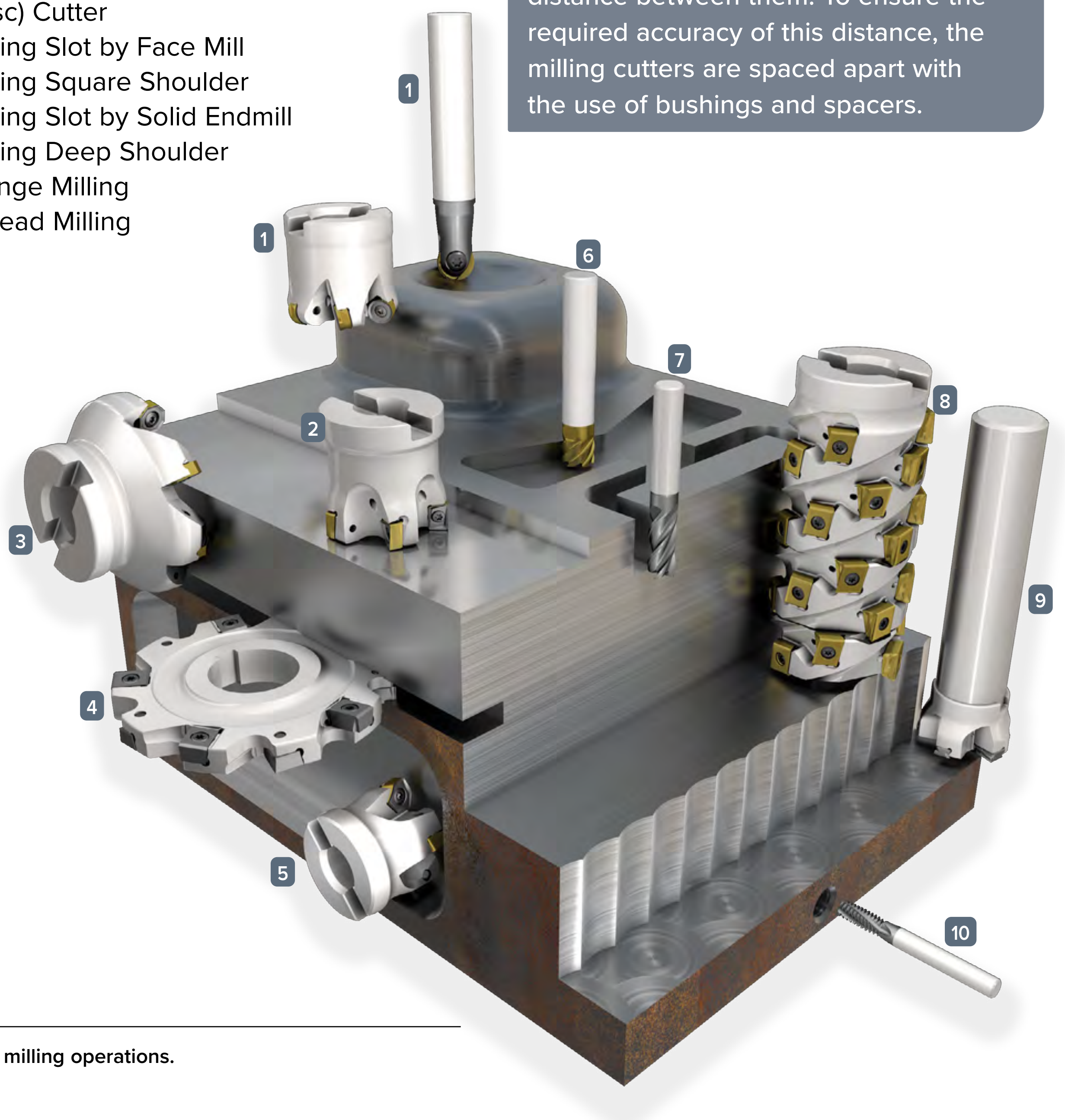


Fig. 8. Main milling operations.

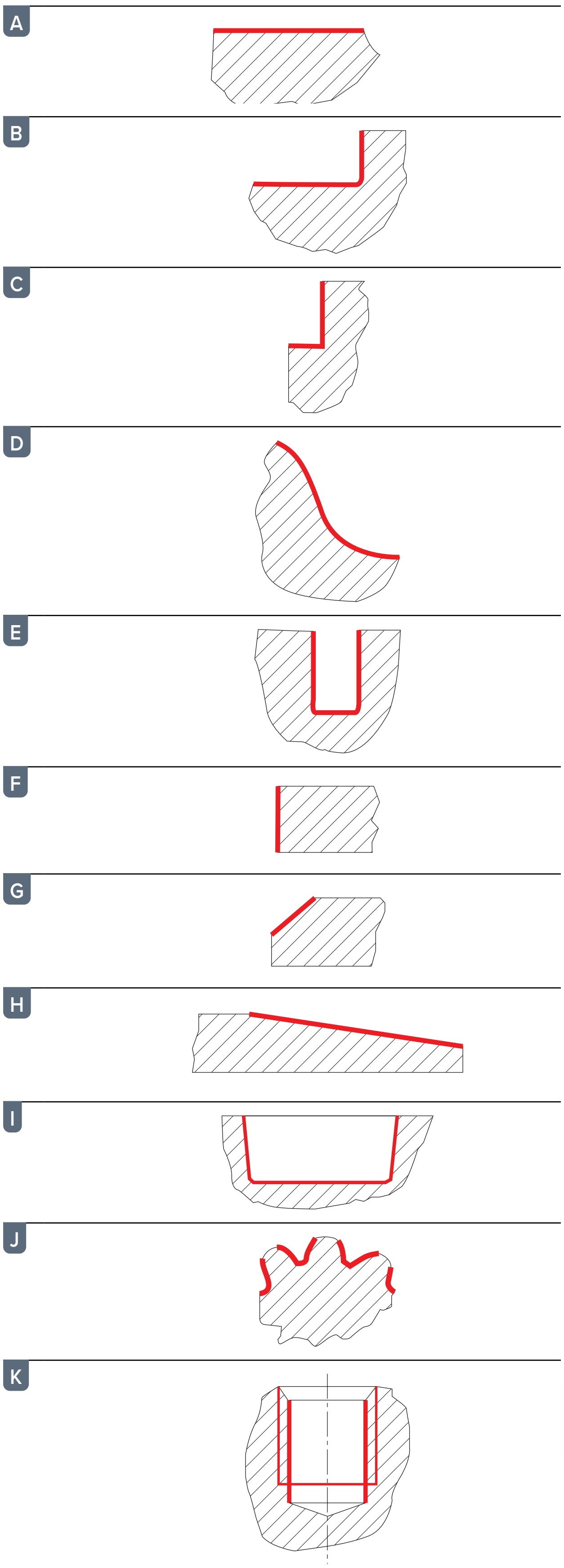


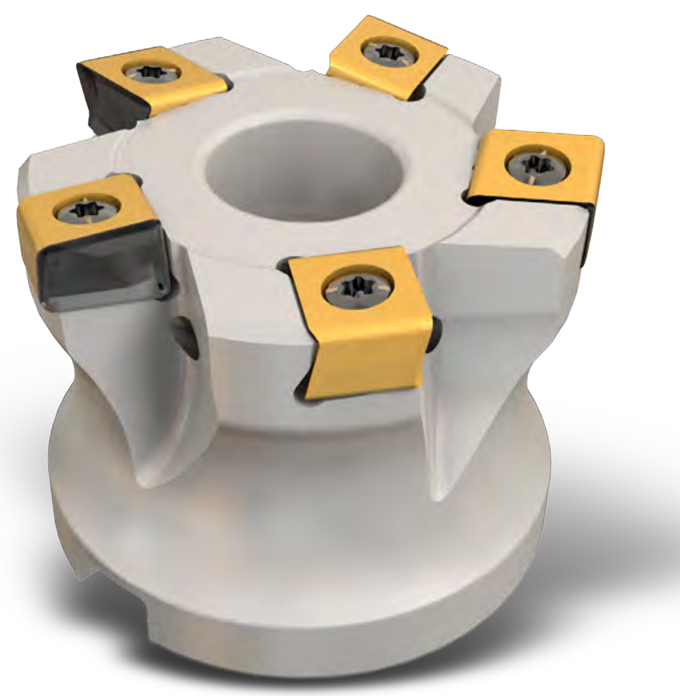
Fig. 9. Illustration of surfaces machined using primary milling operations.

The machined surfaces are highlighted in red.

A — Milling open flat face.
B — Milling flat face bounded by square shoulder.
C — Milling square shoulder.
D — Milling 3D surface (profiling).
E — Milling slot/groove.
F — Milling workpiece edge (edging).
G — Milling chamfer.
H — Milling inclined flat surface.
I — Milling pocket/cavity.
J — Gear/spline milling.
K — Milling inner thread.

Facing, Profiling, Shouldering

In turning, the definitions “facing”, “profiling”, and “shouldering” are used to specify typical turning operations. However, in milling, these words are considered “shop talk”, replacing the full terms “face milling”, “profile milling”, and “shoulder milling”.



Up (Conventional) and Down (Climb) Milling

Depending on the interrelation between the primary and feed motions, there are two types of milling: up and down milling.

In up (up-cut, conventional) milling (Fig. 10), the tool rotation (primary motion) with a rotational velocity of n is directed against the feed motion, which has a linear speed of v_f. Each tooth of the tool gradually cuts into the material, starting with a very thin chip and increasing the chip thickness progressively until it reaches its maximum h_{max} at the tooth's exit from the material.

Up and Down

To explain the origin of terms "up" and "down" that characterize the types of milling, let's examine peripheral milling on a horizontal milling machine. In this case, a cylindrical mill is used as the cutting tool, and a workpiece is mounted on the machine table, as shown in Fig. 2. In up milling, the cutting force exerted by the mill tries to move a machined workpiece up from the table. On the other hand, in down milling, the cutting force pushes the workpiece down onto the table.

In down (down-cut, climb) milling (Fig. 11), the primary and feed motions have the same direction. Each tooth starts cutting with a maximum chip thickness that gradually decreases to a very small value at the tooth's exit.

Despite down milling being the preferred method of cutting due to its better conditions for effective cutting and higher surface finish, up milling does have its advantages in specific cases. For example, up milling is often favored for rough milling hard-skin castings or scaly forgings.

Pure up or down milling is observed only when the tool axis is located outside the material being machined. When the axis lies within the material, the tool teeth are engaged in both types of milling (Fig. 12, areas I and II). This combination is typical in most face milling applications.

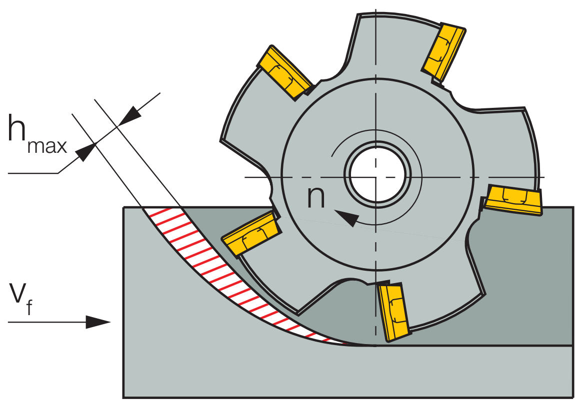


Fig. 10. Up (up-cut or conventional) milling.

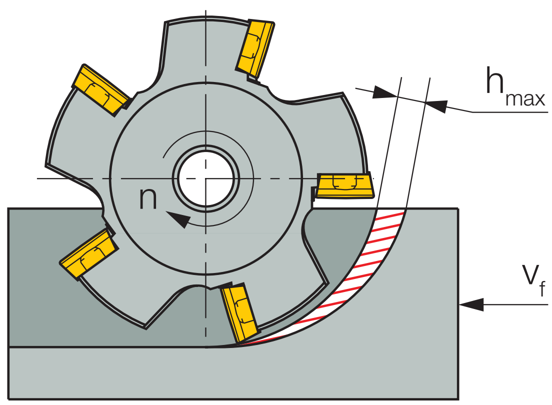


Fig. 11. Down (down-cut or climb) milling.

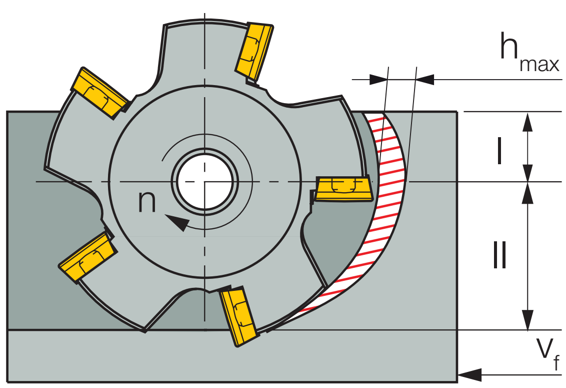


Fig. 12. Combination of up and down milling when the axis of the tool lies within the material being machined. Area I features up milling, while area II is characterized by down milling.

Self-evaluation quiz

1- **What is the primary motion in milling?**

a. Rotary motion of a mill.

b. Rotary motion of a workpiece.

c. The primary motion can be utilized with both above motions.

d. Linear motion of a workpiece.

2- **The feed motion in milling is**

a. The linear motion of a machined workpiece.

b. The rotary motion of a machined workpiece.

c. The translational motion of a machined workpiece relative to the cutter.

d. The rotary motion of a mill.

3- **The cylindrical mill is intended for**

a. Peripheral milling.

b. Face milling.

c. End milling

d. Both peripheral and face milling.

4- **Side-and-face milling relates to**

a. Milling plane face bounded by shoulder.

b. End milling.

c. Milling slots and groove by disc milling cutter.

d. Milling complex 3D surfaces.

5- **Should up milling, which ensures a better cutting effect, be considered as the first-choice type of milling, and applied wherever possible?**

a. Yes, it is correct.

b. No, it is not correct.

6- **Is the combination of up and down milling typical for most face milling operations?**

a. Yes, it is correct.

b. No, it is not correct.

Classification of Milling Cutters

Milling cutters can be classified in different ways based on the following attributes:

1- **Milling method:** Slab mills, face mills, and endmills are examples of milling tools that are intended for milling by different methods.

2- **Type of machined surface:** Plane, shoulder, 3D surface are typical surfaces to be milled.

3- **Main usage:** Face mills, cylindrical endmills, slot mills, profile mills, thread mills, chamfer mills, form gear mills, and corner rounding mills serve distinct purposes, while high feed mills, high-speed mills, and trochoidal mills have specific applications.

4- **Direction of rotation:** Milling cutters can be classified as right-hand (RH) or left-hand (LH).

5- **Machining type:** General-duty, rough, and finish mills address different machining needs.

6- **Cutting part profile:** Cylindrical, tapered, toroidal, ball nose, circle-segment, back draft, and disc shapes are common cutting part profiles.

7- **Design concept:** Milling cutters can be solid or assembled. Assembled tools feature a tool body carrying a cutting part formed by elements mounted on the body, such as inserts with indexable cutting edges.

8- **Material of cutting part:** High-speed steel (HSS), cemented carbides, polycrystalline diamond (PCD), and ceramics are commonly used materials for the cutting part.

9- **Mounting method:** Arbor-type shell mills have a central bore for mounting on an arbor, while shank-type mills have a shank for clamping in a holder.

10- **Adjustability:** Milling cutters can be adjustable, with an adjustable cutting part, or non-adjustable without such capability.

11- **Destination:** Milling cutters can be standard or customized based on specific requirements.

Figures 13 and 14 provide examples of milling cutters with their description and appropriate attributes.

Historical Notes: The Milling Cutter of Eli Whitney

There is no consensus on who exactly invented the milling cutter. However, many give credit to Eli Whitney, an American inventor and manufacturer, who made significant contributions to the concept of interchangeable parts. This concept marked an early sign of mass production. In the late 18th century, Eli Whitney designed an innovative multi-blade rotary tool, resembling an iron wheel with cutting teeth on its periphery – a precursor to modern milling cutters.

Fig. 13. An assembled right-hand face mill with indexable inserts. (This general-duty mill is primarily used for machining open planes. It features a shell-mill configuration and is equipped with indexable carbide inserts).

Fig. 14. A solid carbide ball nose endmill. (The ball-nose shape of this specific profile end mill makes it perfect for machining 3D surfaces, especially in semi-finish to finish operations. It is designed with a shank-type structure and produced from tungsten carbide).

Standard Tools

The definition “standard tool” has a certain duality. On the one hand, it may mean that a tool meets the requirements of a national (international) standard. On the other hand, cutting tool manufacturers use this definition to specify their in-stock products of standard delivery.

Weldon

In milling tools, the term “Weldon” designates the cylindrical shank of a tool (usually a milling cutter) with one or two side flats for clamping and driving. This type of shank was originally introduced by Weldon Tool Co. in the 1920s.

The Main Elements of a Milling Cutter

As with any cutting tool, a milling cutter consists of two main elements: a body and a cutting part. The cutting part directly performs the cutting action, while the body is responsible for mounting the cutter and transmitting the necessary torque from the main drive of a milling machine tool. The body also aids in chip evacuation, facilitating the removal of chips produced during cutting. In modern milling cutters, the body may include inner channels for coolant supply to the cutting zone, making it a cutter with an inner coolant supply option.

Why "mill"?

The term "milling cutters" being named "mills" actually originates from the verb "to mill", which means to grind or reduce to fragments - the process of material using a milling machine. This association probably comes from the similarity between the cutting action performed by milling cutters and the grinding or milling process used in early grain mills, where grain was ground or milled to produce flour. Therefore, the term "mill" was derived from the verb "to mill" and its connection to the grinding or milling action performed by these cutters. Some people believe that the term "mill" in the context of milling cutters originated from the design resembling a classical windmill. While the design may not directly evoke windmills, it is an interesting analogy to consider.

The cutting part of a mill is represented by its teeth. In assembled mills, the teeth are constituted by separate components that are mounted on the body. In contrast, solid mills have the cutting part formed by a specially shaped area of the body.

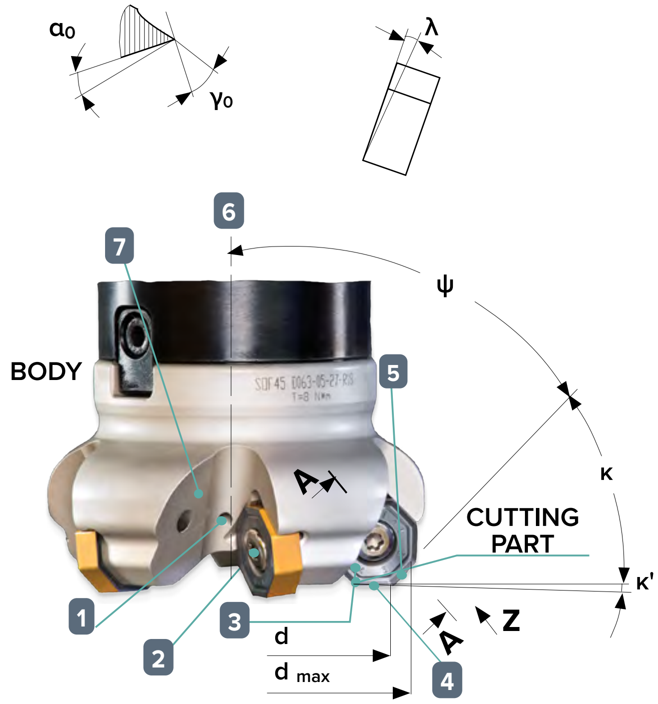
The body of arbor-type mills has a bore for mounting a mill on an arbor or a machine spindle, while shank-type mills feature a shank that is necessary for securing a mill in a holder.

Chip ... Gullet, Mouth, Throat, or Slot?

All these terms relate to the area of a cutting tool designed for chip flow during machining. The chip mouth and chip throat are usually shaped holes, and the chip gullet and chip slot are grooves. In rotating tools, the terms “chip mouth” and “chip throat” are more common in hole making, while the terms “chip gullet” and “chip slot” are used more in milling.

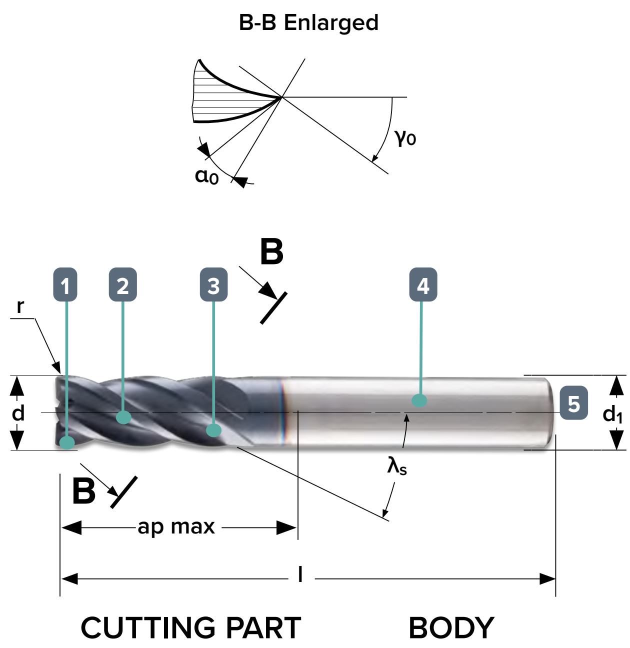
To ensure the smooth flow of chips produced during cutting, a milling cutter features a specially designed chip space or cavity between its teeth. This chip space must be adequately sized to allow for unrestricted chip flow.

There are different terms used to describe such a space in milling, the most common terms being “chip gullet” and “flute”. The term “chip gullet” is generally used to refer to the chip space in indexable milling cutters. Conversely, “flute” is primarily used for solid mill designs, where it denotes a helical groove that facilitates chip flow while also creating a sharp cutting edge or mill tooth along one of its edges.



1 — coolant outlet
2 — insert clamping screw
3 — indexable insert
4 — minor cutting edge
5 — major cutting edge
6 — tool axis
7 — chip gullet

Fig. 15. Main structure elements and geometrical parameters of an indexable milling cutter.



1 — minor cutting edge
2 — flute
3 — major cutting edge
4 — shank
5 — tool axis

Fig. 16. Main structure elements and geometrical parameters of a solid endmill

Table 1. The designation of main geometrical parameters of milling cutters.

Parameter	Common Designation**	Designation Acc.to ISO 13399
Cutting Diameter	d	DC
Maximum Cutting Diameter	d_{max}	DCX
Tool Cutting Edge Angle	κ	KAPR
Tool Minor Cutting Edge Angle	κ'	
Tool Lead (approach) Angle	ψ	
Tool Cutting Edge Inclination	λ	
Tool Orthogonal Rake (rake)	γ_0	GAM...***
Tool Orthogonal Clearance (relief)	α_0	AN...
Flute Helix Angle	λ_s (ω)	FHA
Overall Length	l	OAL
Maximum Depth of Cut	$a_{p\ max}$	APMX
Shank Diameter	d_1	DMM
Corner Radius	r	RE

** Based on ISO 3002/1
***ISO 13399 uses radial and axial rakes

The Geometry of a Milling Cutter

Like any cutting tool, a milling cutter must have appropriate geometry to facilitate the cutting action. The main geometrical characteristics of milling cutters are illustrated in Figures 15 and 16.

Cutting capability and applicability of a mill are mainly determined by the following angles:

- the rake (or rake angle), designated as γ,
- the clearance (also referred to as "relief", "clearance angle", or "relief angle"), designated as α,
- the cutting edge inclination, designated as λ.
- the cutting edge angle, designated as κ,
- the minor cutting edge angle, designated as κ'.

These angles can be determined in different ways depending on the chosen reference system of planes. For instance, if the rake (γ) and the clearance (α) are specified in so-called orthogonal reference plane, they are denoted by adding subscript "o": γ_o and α_o, respectively. The determination of the systems of planes is specified by appropriate normative documents, in particular, by ISO 3002/1 standard.

The angles that define the cutting geometry of a mill have a significant impact on chip formation and chip flow, cutting force, power consumption, and play a crucial role in determining the strength of the mill's cutting edges.

In addition to the nominal value, both the rake angle and the cutting edge inclination have specific directions and appropriate signs. For instance, in Fig. 15, the orthogonal rake (γ_o) is positive, while the cutting edge inclination (λ) is negative.

"Side" and "back" or "radial" and "axial"?

With respect to the designation of the rake measured in different reference planes, the terms "side" and "back" are standard definitions that apply to all cutting tools. Conversely, their alternative designations, "radial" and "axial", are primarily applied to rotating tool nomenclatures, while the standard definitions are commonly used in non-rotating tool terminology.

Now, let's take a closer look at the cutting geometry of the mill depicted in this figure with the use of its appropriate enlarged fragment shown in Fig. 17. The rake (γ) can be measured in different reference planes. Along with the orthogonal reference plane, in which the rake is characterized by the orthogonal rake (γ_o), also known as the effective rake, the rake is resolved into two additional components specified in the following planes: normal to the mill axis and parallel to this axis (sections B-B and C-C in Fig. 17, respectively). The rake measured in these planes is designated as the side or radial rake (γ_f) and the back or axial rake (γ_p). Sometimes, radial rake and axial rake are designated by γ_r and γ_a, too.

Depending on the direction of the side (radial) and back (axial) rakes, different geometries of milling cutters exist. If both of these angles are positive, the cutter geometry is considered double positive. Conversely, if both angles are negative, the cutter features double negative geometry. The combination of a positive back rake and a negative side rake forms a positive/negative or shear-angle geometry. These three geometries reflect typical cutting concepts of milling tool design adopted in metalworking industries. Each geometry has its own advantages and disadvantages concerning cutting forces, power consumption, heat generated during machining, strength of cutting edge, chip flow, and therefore, applicability to milling various groups of engineering materials.

Clarifying Terminology: Positive/Negative or Negative/Positive?

When describing the geometry of a milling cutter, it is crucial to understand which rake, whether radial or axial, corresponds to the terms positive or negative. While this may not matter for cutters with double positive or negative geometry, the phrase "positive/negative" can lead to confusion. In the context of our guide, the first word ("positive") in the definition refers to the axial rake of the cutter, while the second word ("negative") characterizes its radial rake. However, it is worth noting that different technical sources may use the opposite order, which completely changes the meaning of the definition. Since the order of the words is not strictly specified, it is essential to consider the context and relevant data to avoid potential mistakes.

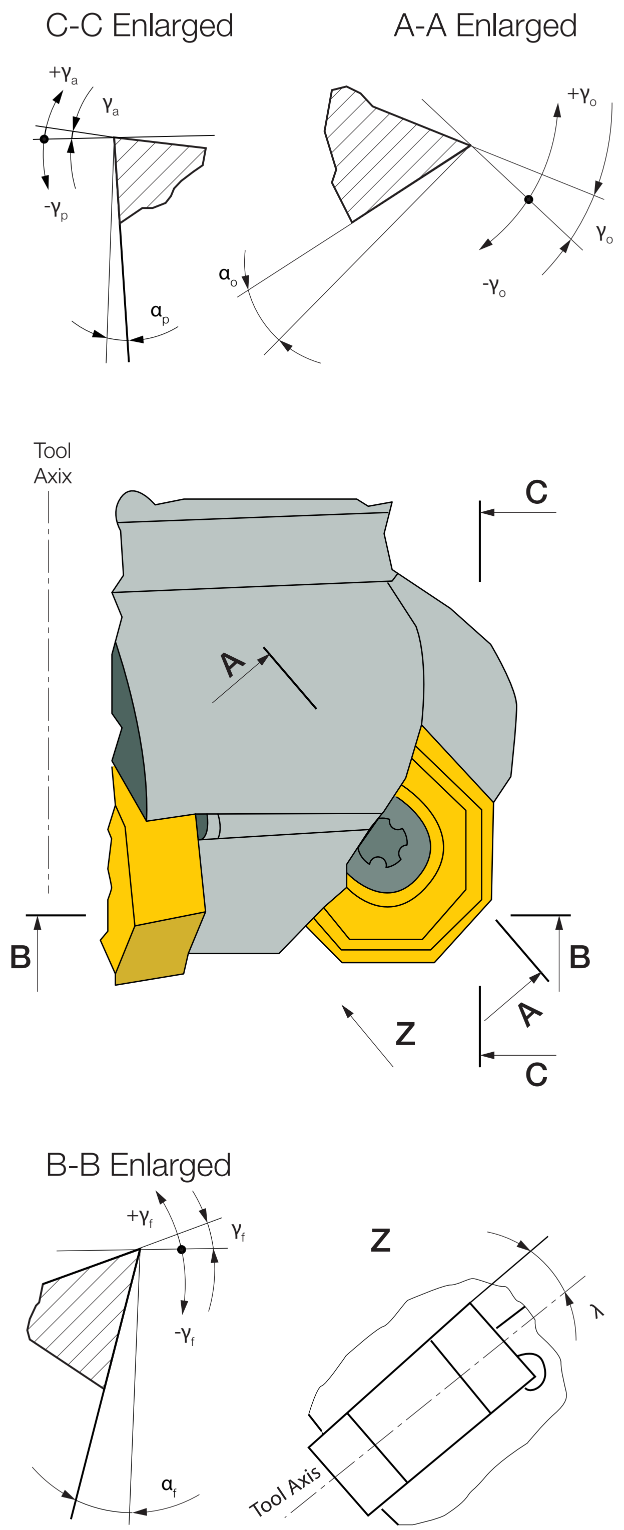


Fig. 17. The rake γ and the clearance α of the indexable milling cutter shown in Fig. 15, each measured in different reference planes.

Neutral Rake and Neutral Geometry

If the rake of a milling cutter is equal to zero, it is often referred to as neutral rake. Milling cutters with both side (radial) and back (axial) rakes equal to zero are designated as cutters with neutral (zero) geometry.

The rake parameters, including value, direction, and sign, are determined by two factors: the cutting edge inclination (λ or λs) and the shape of the mill's rake face. The cutting edge inclination, being an angle, also possesses a specific value, direction, and appropriate sign. For instance, in Figures 15 and 17, the indexable mill exhibits a negative λ. In milling cutters with helical flutes such as solid endmills and slab mills, the cutting edge inclination is determined by the flute helix angle (λs), also denoted as ω.

One Simple Rule to Determine the Sign of the Mill's Angle

There is one simple rule to determine the sign of rake and cutting edge inclination in milling cutters. A positive radial rake suggests that the rake face slopes in the same direction as the primary motion, while a negative radial rake angle means that the rake face slopes opposite to the primary motion. Similarly, a negative axial rake and cutting edge inclination occur when the rake face and cutting edge slope towards the mill face, while a positive slope implies the opposite direction.

The cutting edge inclination plays an important role in smoothly entering the edge into the material and reducing the impact load on the mill. Furthermore, this inclination also affects the direction of chip flow and influences the axial component of a total cutting force.

A Little History: Pocket Inclination and Changes in Insert Design

The first indexable inserts for milling cutters had a simple geometric shape with flat top and bottom faces that were parallel to each other. Consequently, the cutting edge inclination and axial rake were solely determined by the inclination of the insert pocket base in relation to the cutter axis. If the pocket was inclined positively, both the cutting edge inclination and axial rake were positive, and vice versa. However, advancements in technology have significantly changed the shape of inserts, making them more complex. As a result, the cutting edge inclination and axial rake do not necessarily align with the axial inclination of the insert pocket base. For instance, in the mill depicted in Figures 15 and 17, the insert pocket base is inclined negatively, but the axial rake (γp) is positive, while the cutting edge inclination (λ) remains negative. Therefore, when referring to designs of indexable mills based on the pocket base inclination, the terms ‘with positively held inserts’ and ‘with negatively held inserts’ are more accurate. The first term indicates that the insert pocket base is inclined positively (Fig. 18a), while the second term indicates a negative inclination (Fig. 18b).

(a)

(b)

Fig. 18. An indexable milling cutter with positively (a) and negatively (b) held inserts.

Milling Dish

The dish is the angular clearance formed by a mill face towards to the mill axis. A dish is defined by the mill's minor cutting edge angle (κ’) - the angle between the minor cutting edge and a plane normal to the axis. Therefore, the minor cutting edge angle (κ’) is commonly known as the “dish angle”. Both terms, “dish” and “dish angle”, are particularly used in solid endmill nomenclature. Solid endmills often adopt a dish-concept design. However, there are also flat bottom endmills available that have a zero dish angle.

The cutting edge angle (κ) is defined as the angle between the main cutting edge of a milling cutter and the plane containing the direction of feed motion. For example, in the case of a typical face milling cutter, the cutting angle is the angle between the main cutting edge and the plane generated by the cutter. Often, milling cutters are designated based on their cutting edge angle. For instance, mills with a cutting edge angle of 45° and 90° are commonly referred to as 45°-mills and 90°-mills, respectively. In milling cutter designs, typical cutting edge angles have nominal values of 10°, 17°, 30°, 45°, 60°, 65°, 75°, and 90°, with 45° and 90° being the most common angles.

Approach to the Entrance - Be Familiar with Terminology

The cutting edge angle is often designated as the ‘entering’ or ‘entrance’ angle, while the lead angle is referred to as the ‘approach’ angle. In the U.S., the term ‘lead angle’ is more commonly used, while in Europe, ‘approach angle’ is often preferred.

The angle complementary to the cutting edge angle is the lead angle (ψ), so the sum of both angles is 90°. If the cutting edge angle is 65°, then the lead angle will be 25°. The cutting edge angle and the lead angle are equal only for 45° milling cutters.

The cutting edge angle of a mill has a significant impact on the chip thickness during machining, and consequently, the feed rate. This angle is a crucial factor in determining the suitability of a cutter. For instance, a mill with a cutting edge angle other than 90° is often unsuitable for machining a face that is bounded by a shoulder. Furthermore, the angle κ, which is a parameter in mill design, also influences the maximum cutting depth and other characteristics of the mill.

Face mills and endmills have an angular clearance made on a mill face towards the mill axis to generate a flat surface. Such a clearance is defined by the minor cutting edge angle (κ') - the angle between the mill's minor cutting edge and a plane normal to the axis.

In addition to the described angles that determine the active cutting part of a mill, there are linear dimensions, which specify the size and configuration of the mill. These dimensions include the overall length, diameters of different areas of the mill, the diameter of the central bore (for arbor-type mills) or the shank (for shank-type mills), maximum depth of cut, flute length (for mills with flutes), and more.

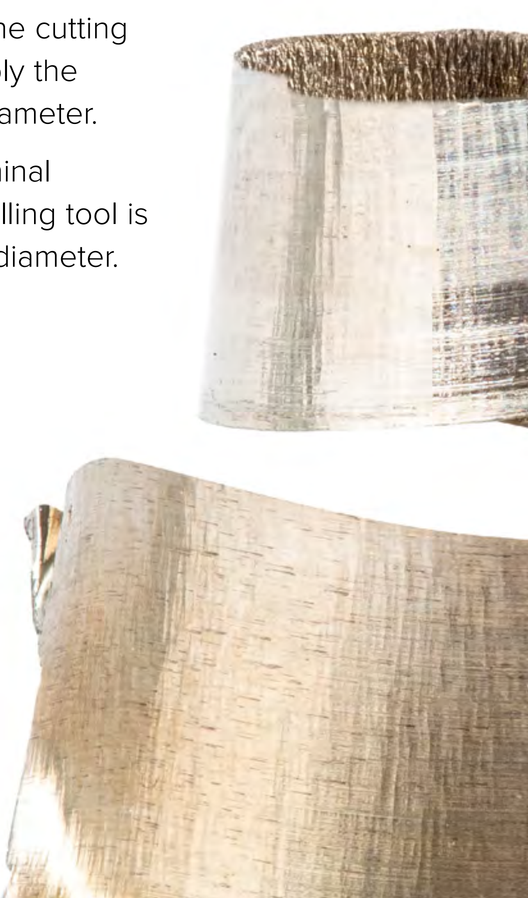
Exploring ...-Degree Milling Cutters

The terms such as "90° milling cutter" and "45-degree face mill" refer to mills with corresponding cutting edge angles. For example, a milling cutter with a 90° cutting edge angle or a face mill with a 45-degree cutting edge angle. It is important to note that in 90° mills, the cutting edge inclination actually represents the axial rake.

Special attention should be given to the cutting diameter (d). For face mills, endmills, and toroidal-shaped mills, the cutting diameter determines the maximum width of the flat surface generated by the mill. According to ISO 6462 standard, the cutting diameter (d) should be measured from point P, as shown in Fig. 19. Point P is the theoretical point where the main cutting edge intersects the plane perpendicular to the mill axis.

For ball-nose and circle-segment mills, the cutting diameter is simply the maximum mill diameter.

Usually, the nominal diameter of a milling tool is the tool cutting diameter.



Clarifying Terminology: Effective Diameter

In some technical publications, the cutting diameter (d) is sometimes incorrectly referred to as the "effective diameter." Strictly speaking, the term "effective diameter" actually refers to the largest true cutting diameter, typically measured at the axial depth of cut. It is important to understand the distinction between these two diameters, especially in profile milling with tools that have a shaped or non-straight form. In such cases, the cutting diameter varies based on the depth of cut and may differ across different areas of the tool's cutting edge involved in the milling process. The effective diameter, on the other hand, represents the largest true cutting diameter, which is the maximum of the cutting diameters of these different areas.

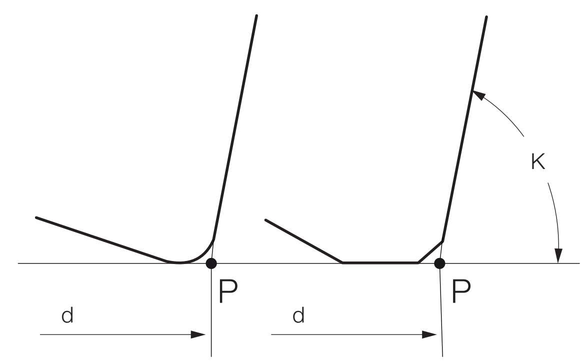


Fig. 19. The cutting diameter (d) in accordance with ISO 6462 standard.

Clarifying Terminology: High Positive Milling Cutter

In cutting tools, the term "high positive" is commonly used to define the various features of tools and indexable inserts, particularly an insert shape, a tool cutting geometry, and others. With respect to the milling cutters, especially the indexable ones, this term usually relates to a tool with extremely positive axial and radial rakes.

Pitch of a Milling Cutter

The pitch of a milling cutter is the distance between the two nearest neighboring teeth of the cutter, measured between the same points on the teeth. The pitch indicates the number of teeth (z) and the space between the teeth that a milling cutter of a specific diameter has. Also, the pitch demonstrates the tooth density of a mill, distinguishing milling cutters with coarse, fine, and extra fine pitches. In addition to the coarse-fine-extra fine pitch rating, alternative gradings such as coarse-regular-fine, normal-close-extra close, and others exist. Moreover, extra-fine- (extra-close-) pitch milling cutters are also referred to as high-density mills.

As an angular dimension, the pitch is represented by the appropriate angle (φ), which is alternatively known as the angular pitch:

$$\varphi = 360°/z \qquad \text{(M1)}$$

In various milling cutter designs, the cutter teeth can be spaced equally or unequally. When the teeth are spaced equally, it is referred to as an equal or even pitch, while uneven spacing is known as an unequal or uneven pitch. The unequal pitch is also termed “differential pitch”, “variable pitch”, or “asymmetrical index” (Fig. 20).

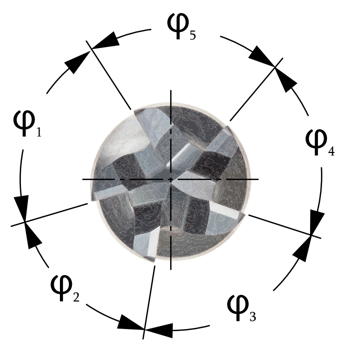


Fig. 20. A solid carbide endmill with differential pitch features unequal teeth spacing, as shown from on-end view.

The partial case of the unequal-pitch design concept is the alternating teeth spacing principle, which can only be applied to milling cutters with an even number of teeth. According to this principle, the change in pitch occurs in pairs, repeating the pitch value of every other tooth (Fig. 21).

For example, a milling cutter with four teeth can have the following design configuration:

- equal teeth spacing with constant 90° pitch,
- unequal teeth spacing with variable angular pitches of 91°, 88°, 89°, and 92°,
- alternating spacing with angular pitches vary as 91°, 89°, 91°, and 89°, with each pair of pitches being identical.

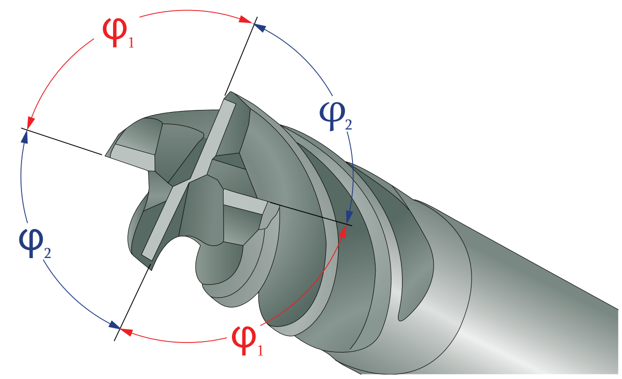


Fig. 21. A solid carbide endmill with alternating teeth spacing has each pair of pitches being identical.

Along with cutting geometry, which plays a pivotal role in determining the cutting capabilities of a mill tooth, tooth density is one of the essential parameters of a milling cutter. When selecting a suitable mill, tooth density greatly impacts the cutter's applicability, depending on factors such as the machined material, required surface finish, and the working characteristics of the available machine tool.

Avoid Misunderstandings: Regular, Even, Spacing, and Indexing

Sometimes, constant teeth spacing is referred to as regular or even. However, this can lead to confusion as "regular" also describes the tooth density of a milling cutter based on the number of teeth it has for a given diameter. Similarly, "even" is used to denote a cutter with an even number of teeth. Additionally, in certain cases, "teeth indexing" is used instead of "teeth spacing". This substitution can also lead to misconceptions, as "indexing'" in cutting tools commonly refers to the ability to reposition a cutting edge in indexable inserts by rotating or reversing the insert. Therefore, when terms with double meanings are used, it is crucial to strictly define them within the context.

A higher number of teeth generally leads to increased productivity. However, as the number of teeth grows, the chip gullet volume naturally decreases. This reduction in chip gullet volume is crucial for sufficient chip evacuation when milling materials that produce different types of chips.

All other factors being equal, using a mill of the same diameter but with a larger number of teeth requires higher cutting power and feed speed. This can pose challenges if the milling machine has limited power or feed drive.

Furthermore, the pitch is a crucial factor that affects the stability of milling operations. In many cases, the use of a variable pitch design enhances the vibration resistance of a milling cutter, leading to improved dynamic behavior of the entire technological system.

Helix Angle

In endmills and slab mills, the solid and brazed carbide cutter design concepts offer a one-piece tool where both the teeth and the chip gullet are formed by a groove. This groove, known as the flute in the solid design concept, can be helical or straight. The helical flute ensures smooth cutting and is therefore more common, while endmills and slab mills with straight flutes have limited usage.

The helix of a flute is determined by two parameters: the helix hand and the helix angle. The helix angle is usually designated by λs or ω. The helix angle of a solid mill is similar to the cutting edge inclination of an indexable or inserted-blade mill. To define the helix hand, you need to look at the mill from its end. In mills with a right-hand (RH) helix, the mill flutes spiral outward from you in a clockwise (CW) direction, while in mills with a left-hand (LH) helix, the flutes spiral outward from you in a counterclockwise (CCW) direction. Since there are both right-hand and left-hand cutting mills available, the following combinations are possible:

- Right-hand cutting mills with a right-hand helix,
- Right-hand cutting mills with a left-hand helix,
- Left-hand cutting mills with a right-hand helix,
- Left-hand cutting mills with a left-hand helix.

The prevailing design is the right-hand cutting mills with a right-hand helix (Fig. 22).

Right or Left: Examining the Mill End

Milling tools can be designed for either right-hand or left-hand cutting. To determine whether a cutter is right-hand or left-hand, one can examine the end of the mill. A right-hand mill should rotate counter clockwise to cut a material, while a left-hand mill would rotate clockwise. It is important to note that the hand of cut does not necessarily correspond to the hand of the helix, so it is crucial to remember this distinction in order to avoid misunderstandings.

The helix hand and helix angle are important factors for the distribution of the total cutting force into components. They also have an impact on the axial force, which affects the loading of a machine spindle and can either press the mill into the spindle or pull it out. Additionally, these factors play a crucial role in facilitating the normal flow of chips during machining. Furthermore, the helix angle also influences the mill rake.

Flute Helix Angle Changes

In addition to mills with a constant flute helix angle, there are mill designs where the helix angle varies. According to the concept of a “variable helix,” different mill flutes feature different helix angles. On the other hand, the “varying helix” design utilizes a helix angle that changes along the flute. Although the terms “variable” and “varying” are often used interchangeably to specify the changeable helix angle of a mill flute, technically, this is not correct.

Fig. 22. A right-hand cutting solid carbide endmill with right-hand helix.

Designing the Geometry of a Milling Cutter

The angles that define the cutting capabilities of a mill, such as the rake, clearance, and cutting edge angle, as well as the mill pitch and helix, are crucial design parameters that greatly influence the mill's performance. Determining these parameters during the mill design depends on the specific application field and aims to provide the optimal milling solution, considering various factors such as the types of materials to be machined, the performed milling operations, and the required maximum depth and width of cut. The field of metal cutting theory and machining practice offers suitable ranges for these angles, enabling the calculation of the strength of mill teeth and estimation of the necessary cross-section of a chip gullet. Additionally, computational mechanics modeling and finite element analysis (FEA) allow for the simulation of cutting action and chip flow, facilitating accurate calculations of strength and rigidity. These tools greatly contribute to the optimization of mill geometry during the design process.

Self-evaluation quiz

7- **What are the main elements of a milling cutter?**

a. A body and a cutting part.

b. A body, a cutting part, and inner channels for coolant supply.

c. A body, a specially shaped area of the body that forms a cutting part, and a specially created chip space between the cutter teeth to ensure the chip flow.

8- **Cutting geometry of a milling cutter is mainly determined by**

a. The cutting edge inclination and the cutting edge angle.

b. The set of appropriate angles.

c. The directions of the radial and axial rakes.

9- **The cutting edge angle is also known as**

a. The approach angle.

b. The lead angle.

c. The dish angle

d. The entering angle.

10- **The lead angle is the angle complementary to**

a. The cutting edge inclination.

b. The approach angle.

c. The cutting edge angle.

d. The minor cutting edge angle.

11- **The flutes of a right-hand cutting solid endmill can be**

a. Helical with a right-hand helix.

b. Helical with a left-hand helix.

c. Helical with right- or left-hand helix.

d. Helical or straight but it is more common for designs to feature helical flutes.

Milling Calculations

Cutting Data

The cutting speed (v_c) characterizes the primary motion in milling and refers to the circumferential velocity of the outermost point of the milling cutter's cutting edge.

In the metric system

$$v_c = \frac{(\pi \times d \times n)}{1000} \text{ (m/min)} \quad \text{(M2)}$$

In the U.S. customary system

$$v_c = \frac{(\pi \times d \times n)}{12} \approx \frac{(d \times n)}{(3.82)} \text{ (sfm)} \quad \text{(M3)}$$

where

d – the milling cutter's diameter (mm in equation M2 and inches in equation M3),

n – the rotational velocity of a milling cutter, rpm.

Since, in most cases, a milling cutter rotates at the same velocity as the machine spindle, the rotational velocity (n) is usually associated with the spindle speed.

The cutting speed (v_c) depends on various factors, including the machined material, cutting material, machining stability, cutter engagement, and others. The recommended ranges provide basic values for the cutting speed based on the cutting material and material being machined. The influence of other factors can be estimated using coefficients. Therefore, in milling calculations, the cutting speed is typically considered a given parameter, and it is necessary to determine the corresponding rotational velocity (n):

In the metric system

$$n = \frac{(1000 \times V_c)}{(\pi \times d)} \text{ (rpm)} \quad \text{(M2a)}$$

In the U.S. customary system

$$n = \frac{(12 \times V_c)}{(\pi \times d)} \approx \frac{(3.82 \times V_c)}{d} \text{ (rpm)} \quad \text{(M3a)}$$

The feed speed (v_f) – the speed with which the milling cutter moves relative to a machined workpiece – specifies the feed motion. This speed that is also referred to as "feed rate", "minute feed", and "table feed" is a function of the strength of the milling cutter's tooth. Strength defines the ultimate load which a tooth can carry. Therefore, a maximum speed feed on conversion to one tooth exists. On the other hand, there is a minimum for such a feed speed to enable cutting action and to avoid deformation of a machined material. The feed speed in tooth equivalent is known as feed per tooth (f_z). Namely this feed, which depends on machined and cutting materials, and the milling cutter's design, is the given parameter for feed speed calculation. Feed per tooth reflects the movement of one tooth in the direction of the feed motion per one revolution of the cutter. Accordingly, the cutter itself will move during one revolution in this direction by an amount called feed per revolution (f or f_r).

The feed speed (v_f), which refers to the speed at which the milling cutter moves relative to the machined workpiece, determines the feed motion. It is also known as the "feed rate", "minute feed", "table feed", or "feed rate". The feed speed is influenced by the strength of the milling cutter's tooth, which defines the maximum load it can bear. As a result, there is a maximum feed speed per tooth that must be considered. Conversely, there is a minimum feed speed required to ensure cutting action. The feed speed per tooth, known as feed per tooth (f_z), is a given parameter for feed speed calculation.

This value depends on the machined and cutting materials, as well as the design of the milling cutter. It represents the movement of one tooth in the feed direction per one revolution of the cutter. Consequently, during one revolution, the cutter itself will move by an amount called feed per revolution or simply feed (f_r or f) in the feed direction.

The feed speed (v_f), feed per tooth (f_z), and feed (f) are measured in mm/min, mm/tooth, and mm/rev in the metric system, and in ipm, ipt, and ipr in the U.S. customary system, respectively. These feed types are interconnected through the following relationships:

$f = f_z \times z$ (mm/rev, ipr) **(M4)**

$v_f = f \times n = f_z \times z \times n$ (mm/min, ipm) **(M5)**

Example. When machining an annealed low alloy steel using a 100 mm (4”) in diameter indexable face milling cutter with 7 teeth, the recommended values for cutting speed and feed per tooth are 180 m/min (590 sfm) and 0.2 mm/tooth (.008 ipt), respectively. Find the required spindle speed and feed speed to be set in CNC program.

Number of Teeth or Number of Inserts?

It’s important to note that in indexable milling cutters, the number of teeth and the number of inserts may not be the same. In the case of indexable extended flute milling cutters, for example, the cutter tooth consists of a series of inserts that are individually mounted one-by-one. As a result, the number of inserts will be greater than the number of teeth.

From the equations (M2a) and (M3a):

$$n = \frac{(1000 \times V_c)}{(\pi \times d)} = \frac{(1000 \times 180)}{(\pi \times 100)} = 573\ (rpm)$$

$$n' = \frac{(12 \times V_c)}{(\pi \times d)} = \frac{(12 \times 590)}{(\pi \times 4)} = 563\ (rpm)$$

From the equation (M5):

$$v_f = f_z \times z \times n = 0.2 \times 7 \times 573 = 800.8\ (mm/min)$$

$$v_f' = f_z' \times z \times n' = .008 \times 7 \times 563 = 31.5\ (ipm)$$

In the calculations of cutting speed and feed, there are two aspects that should be taken into consideration.

1- In profile milling, due to the shaped, non-straight form of the tool, a cutting diameter is a function of a depth of cut; and it is not the same for different areas of the tool cutting edge that is involved in milling. The effective diameter de is the largest diameter of a profile milling tool that is engaged in cutting. For example, when machining with a ball-nose endmill at depth of cut that is less than the endmill's radius, this diameter is smaller than the endmill's nominal diameter (Fig. 23). In calculating cutting data, it is very important to consider the effective diameter, because the real cutting speed relates to the effective diameter, while the spindle speed usually refers to the nominal diameter of a tool.

2- There are milling cutters in which the number of teeth on the cutter face is smaller than on the cutter periphery. When such a cutter is used for face milling, the teeth on the face produce the effective number of teeth ze that should be considered in the feed calculation. Conversely, if the feed speed is defined based on the number of peripheral teeth, the teeth on the face will experience significant loading, which can potentially lead to their failure.

How to Remove Doubts: Width or Depth

In peripheral (slab) milling, when a milling cutter removes material using a significant portion of its peripheral cutting edge, the width of cut is commonly defined as the length of this edge, measured along the mill axis. However, this definition can lead to misunderstandings. Similar situations may take place in milling a slot by a disc mill. To avoid confusion, it is recommended to use precise terminology such as “axial depth of cut” and “radial depth of cut”. These definitions provide clear distinctions and remove any doubts.

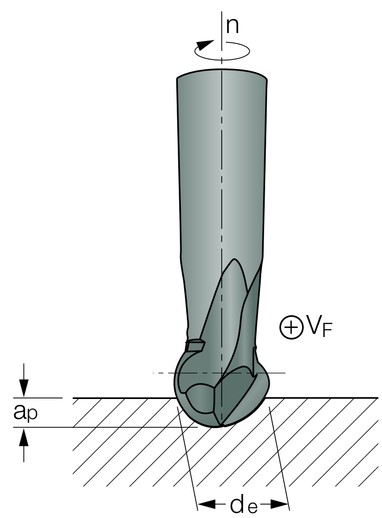


Fig. 23. The effective diameter d_e of a ball-nose endmill.

A milling cutter is used to remove a layer of material with specific geometrical characteristics. The depth of cut (a_p or d.o.c.) refers to the thickness of the material layer that is removed in a single milling pass. The depth of cut is measured along the axis of the cutter, making it synonymous with the axial depth of cut. It represents the difference in height between the machined and non-machined surfaces along the axial direction. On the other hand, the width of cut (a_e or w.o.c.) determines the size of the material layer removed in a single milling pass measured radially. It is also known as the radial depth of cut.

Cutting Conditions

When determining cutting data, the cutting conditions that define the environment in which milling occurs can be a crucial factor. The cutting conditions primarily reflect the stability of the entire machining system, including factors such as tool and workpiece holding, tool overhang, workpiece rigidity, and impact load on the tool. Unfavorable cutting conditions often require a significant reduction in cutting data.

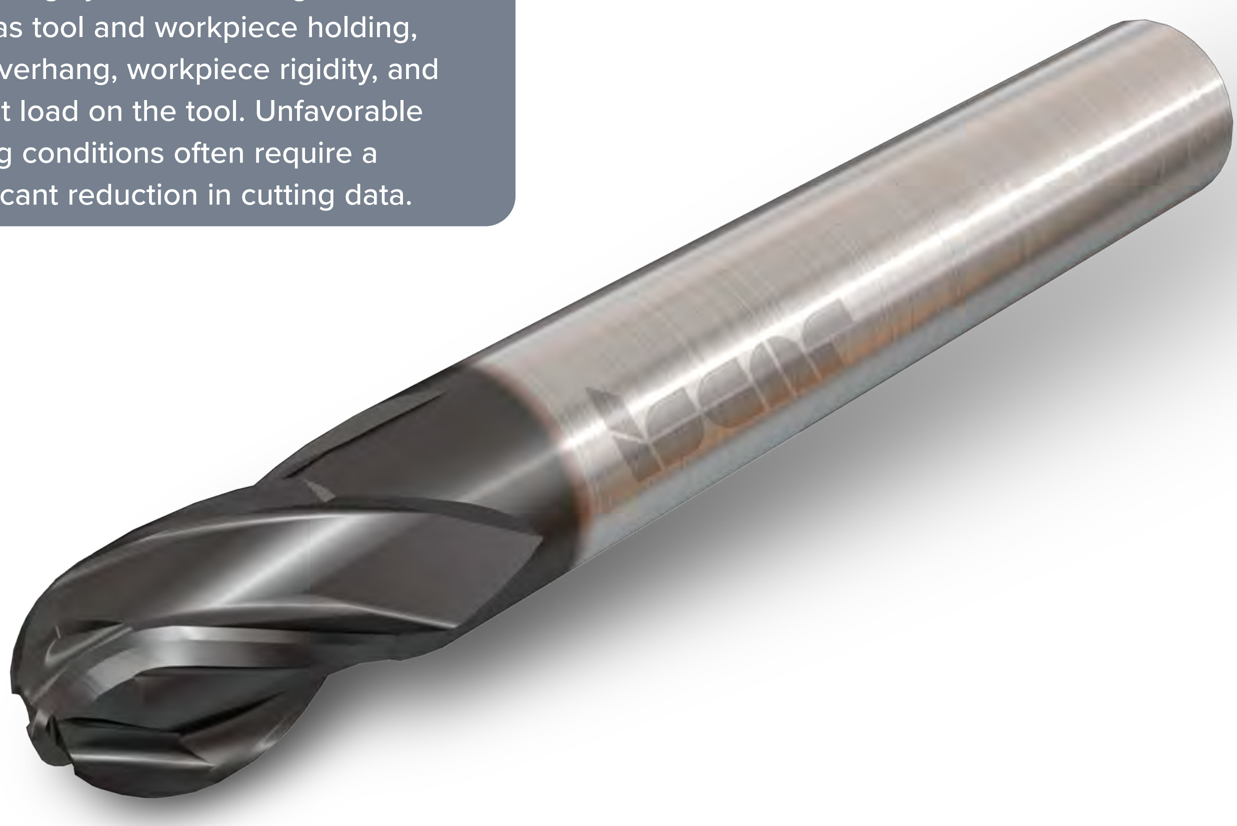
The productivity of a milling operation is measured by the metal removal rate or MRR (Q), which represents the volume of material removed by the cutter per unit of time.

$$Q=a_p \times a_e \times v_f=f_z \times z \times n \quad \textbf{(M6)}$$

Example. Find metal removal rate for the milling cutter in the previous example when the cutter is operated with the same cutting speed and feed per tooth, and with 5 mm (.2”) depth of cut and 76 mm (3”) width of cut.

$Q= a_p \times a_e \times v_f = 5 \times 76 \times 800.8 = 304304\ (mm^3) = 304.3\ cm^3$

$Q' = a_p \times a_e \times v_f = .2 \times 3 \times 31.5 = 18.9\ (in^3)$



Chip Thickness

The chip thickness (h) reflects the mechanical load on a cutting edge. This thickness has understandable upper and lower boundaries: on the one hand, it should not be so high as to destroy the cutting edge, and on the other hand, not so small as to hinder cutting action. Therefore, maintaining an optimal chip thickness is key to a successful milling operation. Often, the chip thickness is considered to be identical to the feed per tooth (f_z). While in some cases these two parameters can be equal, they are not fundamentally the same. The chip thickness is a function of the feed per tooth; however, there are several factors that can cause chip thinning, thereby reducing the chip thickness relative to the feed per tooth.

The first factor is the cutting geometry of a milling tool, specifically, the tool cutting edge angle (κ). Let's examine Fig. 25. The feed per tooth (f_z) reflects the relative movement of the milling tool with respect to the workpiece as the tool rotates by one tooth. During this movement, the tool removes a layer of material, producing a chip. In the direction of the axis of a milling cutter, the cross-section of the chip features the maximum chip thickness (hmax). In the case of 90° milling tools, the maximum chip thickness is equal to the feed per tooth (Fig. 24a). However, if the cutting edge angle is different from 90°, the generated chip is thinner, and the maximum chip thickness is smaller than the feed per tooth (Fig. 24b). Table 2 shows reducing the maximum chip thickness as a function of the cutting edge angle. In case of profile milling tools, particularly ball-nose endmills (Fig. 25), the maximum chip thickness depends on the axial depth of cut (a_p). This reduction in chip thickness in the direction of the mill axis due to the cutting geometry of a milling tool and the axial depth of cut is known as axial thinning.

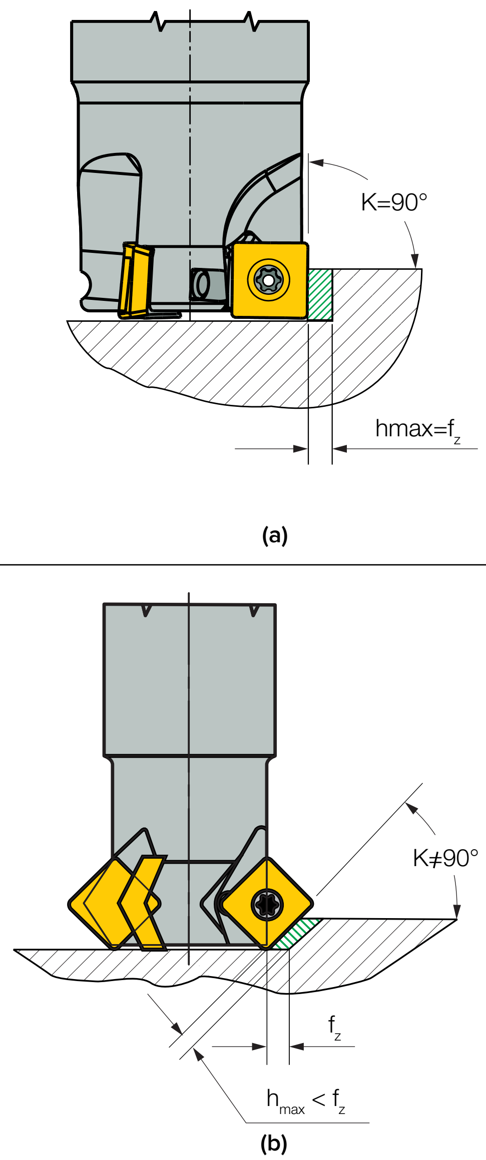


Fig. 24. The maximum chip thickness (hmax) with respect to the feed per tooth (f_z) and the cutting edge angle (κ).

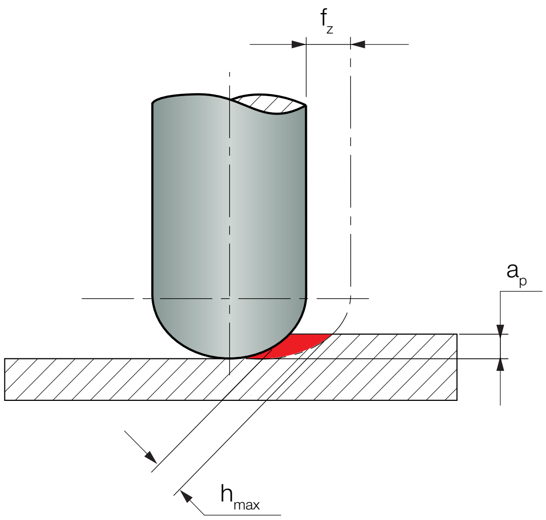


Fig. 25. Axial chip thinning in milling with a ball nose milling cutter.

‘Chip Load’

The term “chip load” is often used as a synonym for the term “feed per tooth”. This term is more commonly used in the North American market. However, the correct synonym for “chip load” is “chip thickness”, specifically referring to the maximum chip thickness.

Table 2. The maximum chip thickness for the same feed per tooth as a function of the cutting edge angle.

Cutting edge angle (κ)	90˚	75˚	60˚	45˚	30˚	17˚	10˚
Lead angle (ψ)	0˚	15˚	30˚	45˚	60˚	73˚	80˚
Maximum chip thickness (hmax)	100%	97%	87%	71%	50%	29%	17%

The second factor of chip thinning is influenced by the positioning of a milling cutter in relation to the machined workpiece. The contact between the mill and the workpiece occurs through an arc, which is measured by the angle of engagement (AE). It is important to note that this angle and the width of cut (radial depth of cut) (a_e) are interconnected parameters (Fig. 26).

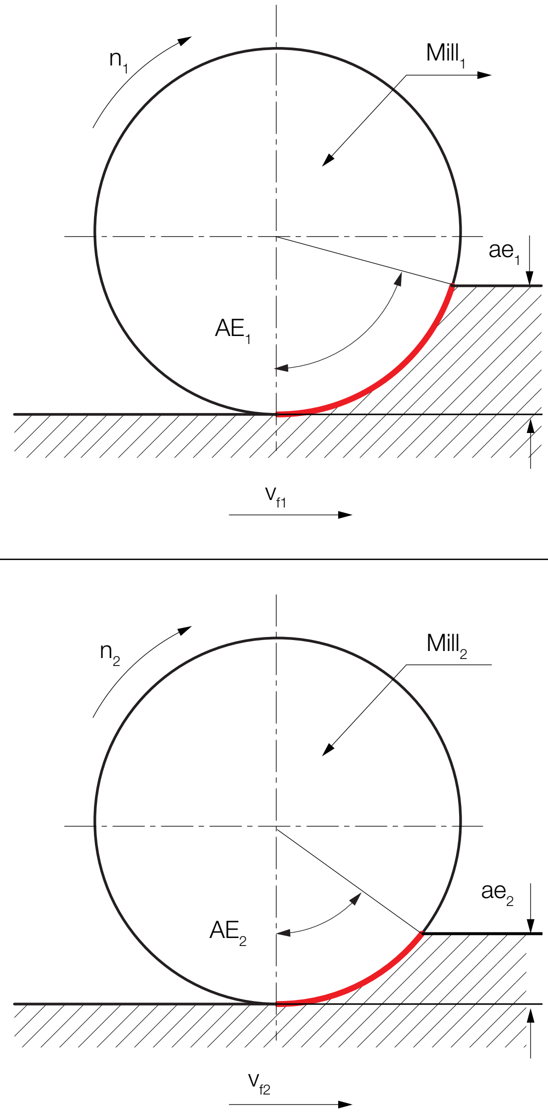


Fig. 26. Contact arc, angle of engagement AE, and width of cut (radial depth of cut) a_e.

Turning our attention to Fig. 27. The thickness of a chip varies along the contact arc in the radial direction, ranging from minimum to maximum. When the radial engagement of the cutter, denoting the width of cut (a_e), reaches half of the cutter diameter, the maximum chip thickness attains the feed-per-tooth value. Conversely, if the radial engagement is less, the maximum chip thickness decreases accordingly. This phenomenon, which refers to the reduction of chip thickness in the cross-section perpendicular to the mill axis, is known as radial chip thinning.

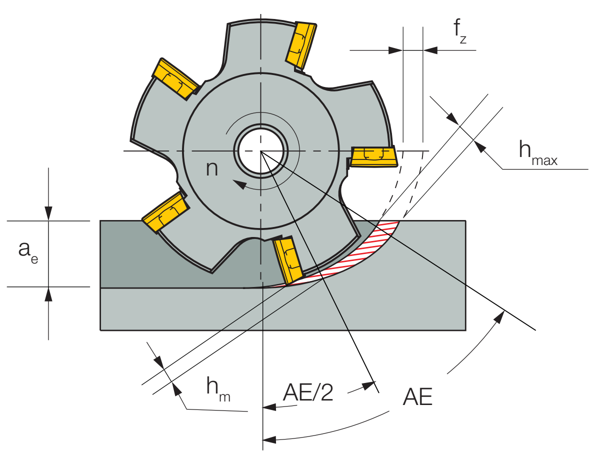


Fig. 27. Chip thickness as the function of a width of cut (radial depth of cut) a_e.

Average Chip Thickness

In many cases, the medium chip thickness (hm) is used as a computed parameter for calculating the feed per tooth (f_z). There are different methods for determining the average chip thickness, such as arithmetic or weighted mean, etc. Sometimes, hm is roughly estimated as half of the maximum chip thickness (hmax). In face and end milling (Fig. 27), the average chip thickness is often calculated in relation to half of the angle of engagement (AE).

Thus, in face and end milling, if the center of a milling cutter in the plan view lies outside the workpiece, the maximum chip thickness (hmax), as the result of radial chip thinning, will be less than the feed per tooth (f_z). The same situation is observed in peripheral (slab) milling and milling slots with disc (side-and-face) cutters when the radial depth of cut is smaller than the cutter radius and the cutter axis is beyond the machined surfaces.

Fig. 28 and Table 3 give the example of calculating maximum and average chip thickness when milling by 90° face mills.

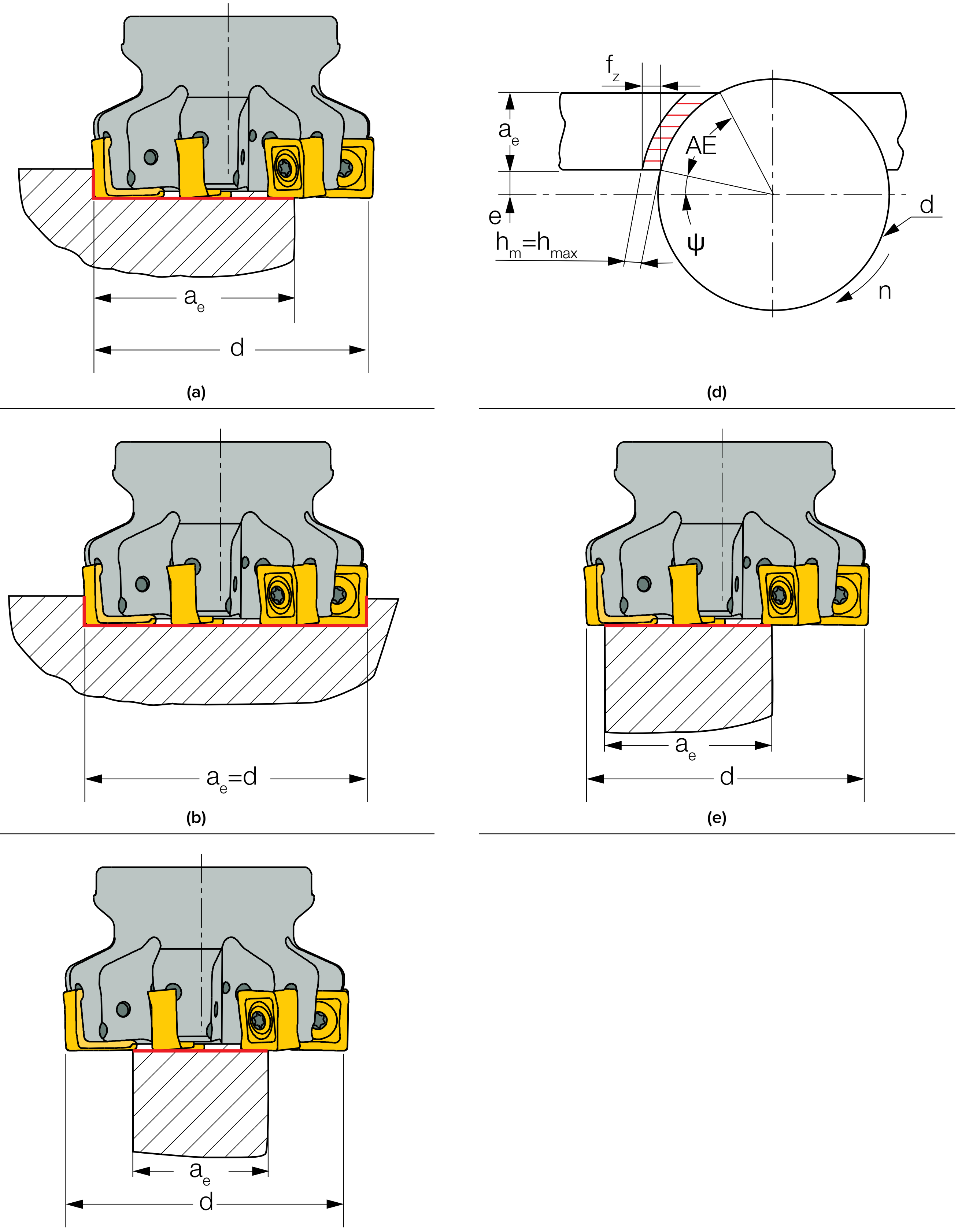
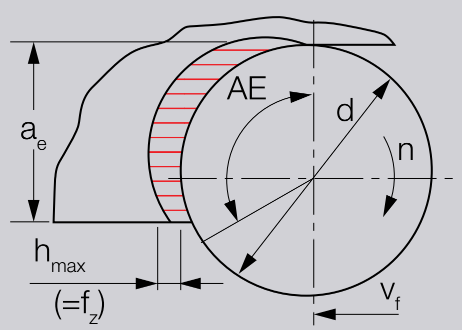


Fig. 28. Milling operations performed by 90° face mills.

Table 3. Calculating chip thickness in face milling.

Fig. 28 Case (a)

Cutter Position

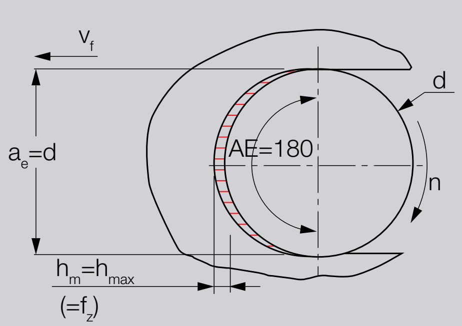


h_m, h_{max}

1st method
$h_m = f_z \times \sin(AE/2)$
$h_{max} = f_z$
2nd method
$h_m = f_z\ 2 \times (\sqrt{2}/2 + \cos(AE-90°)/2)$
$h_{max} = f_z$

Fig. 28 Case (b)

Cutter Position

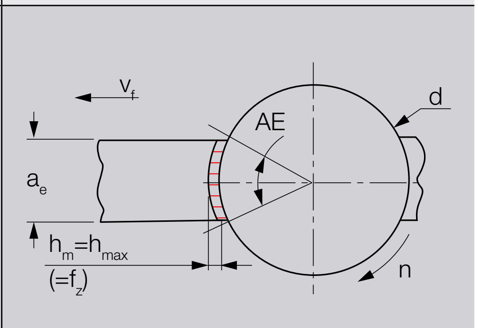


h_m, h_{max}

$h_m = h_{max} = f_z$

Fig. 28 Case (c)*

Cutter Position



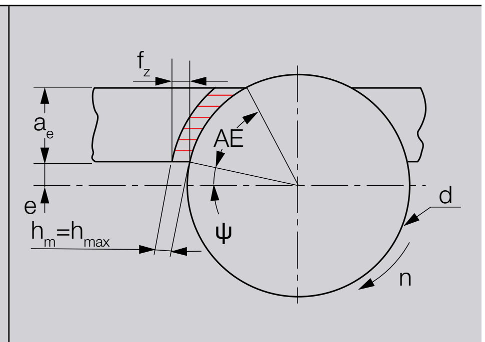
h_m, h_{max}

$h_m = h_{max} = f_z$

* Unfavorable milling conditions - this cutter position should be avoided whenever possible

Fig. 28 Case (d)

Cutter Position

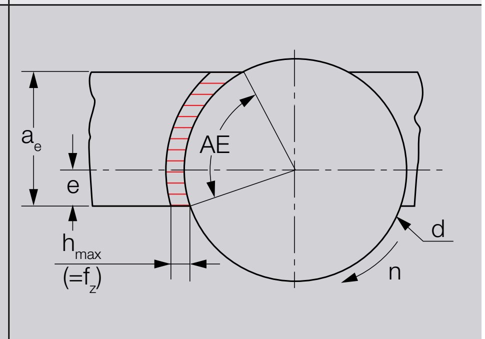


h_m, h_{max}

$h_m = f_z \times \cos(\Psi + AE/2)$
$h_{max} = f_z \times \cos\Psi$

Fig. 28 Case (e)

Cutter Position



h_m, h_{max}

1st method
$h_m = f_z \times \sin(AE/2)$
$h_{max} = f_z$
2nd method
$h_m = h_{max} = f_z$

* Unfavorable milling conditions - this cutter position should be avoided whenever possible

Example. Find the feed per tooth required to achieve an average chip thickness of 0.08 mm (.0031") when milling a square shoulder on a steel workpiece using a 32 mm (1.25") endmill. The shoulder width is 8 mm (.315"), and the milling operation is completed in a single pass.

The width of the cut (a_e) is smaller than the cutter radius, and the operation follows the configuration shown in Fig. 27. Assume that the average chip thickness (hm) is half of the maximum chip thickness (hmax). Hence, hmax=0.16 mm (.0062”).

From Fig. 27, hmax = f_z×sinAE.
AE = arccos ((r-a_e)/r), where r- the cutter radius (16 mm or .625”).
AE = arccos ((r- a_e)/r) arccos ((16-8)/16) = 60°.
Therefore, f_z = hmax/sinAE = 0.16
sin 60° = 0.185 (mm/tooth).

In the U.S. customary system:
AE = arccos ((r- a_e)/r) arccos ((.625-.315)/.625) ≈ 60°.

f_z = hmax/sinAE = .0062/sin60° = .0072 (ipt)

The example demonstrates that in this scenario, in order to achieve the desired average chip thickness, the programmed feed should be 16% higher than the maximum chip thickness value. This emphasizes once again that the feed per tooth and the maximum chip thickness are generally not equal.

Chip Thinning Effect

ISCAR’s Radial Chip Thinning Calculator in Milling is one of the software applications available in the NEOITA engineering calculator library. This tool allows for fast and accurate calculation of the programmed feed per tooth, considering the chip thinning effect. By utilizing this calculator, optimal machining conditions can be easily achieved.

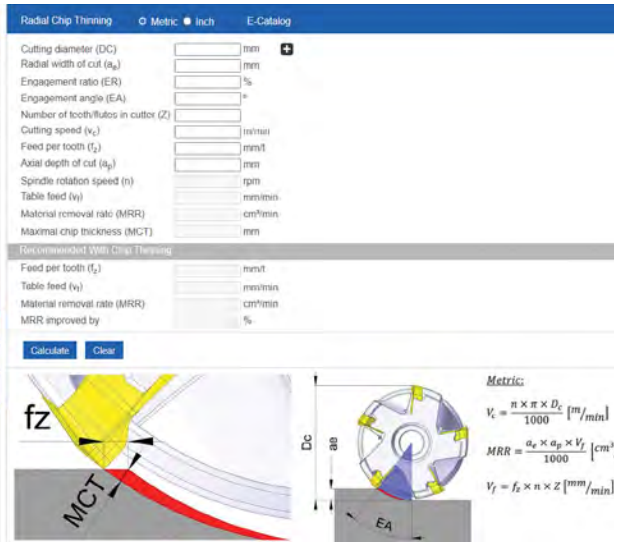


Fig. 29. Radial Chip Thinning Calculator’s data-entry/result screen.

Chip thinning occurs when the maximum chip thickness (hmax) is reduced compared to the feed per tooth (f_z). There are two factors that contribute to this decrease:

- The cutting geometry of a milling tool, particularly the tool cutting edge angle (κ), which is less than 90° (“axial chip thinning”),
- The width of cut (the radial depth of cut) (a_e), when it is smaller than the radius of the milling tool (“radial chip thinning”).

To summarize the brief discussion about chip thinning, whether it is axial chip thinning or radial chip thinning, the more correct way to enhance the milling process is to maintain an optimal chip thickness through an appropriate recalculation of the programmed feed. Chip thinning is essential for comprehending advanced milling techniques, such as high feed milling (HFM) and high-speed milling (HSM).

Cutting Forces, Cutting Torque, and Power Consumption

During the cutting process, each tooth of a mill that comes into contact with the machined material experiences a counteracting force, which prevents material removal. As a result, these teeth are subjected to appropriate forces. The total (or resultant) cutting force (F) is the vector sum of these forces and is typically resolved into the following three components:

- Tangential or circumferential cutting force (F_t), directed tangentially to the outer contour of a mill,
- Radial cutting force (F_r), acting along the mill radius,
- Axial cutting force (F_a), directed toward the mill axis.

$$F = \sqrt{Ft^2+Fr^2+Fa^2} \qquad \text{(M7)}$$

The force action diagram depends on the type of a mill and the milling method. Fig. 29 illustrates cutting forces acting on a face mill.

In milling, the tangential cutting force (F_t) is responsible for the primary work required to remove machined material. This force is crucial for calculating the necessary torque and power consumption of the machine's main drive, as well as determining the strength of the drive elements. The axial cutting force (F_a) determines the load exerted on the spindle bearings in the axial direction. In addition, this force is used for buckling analysis of the mill, particularly when dealing with high mill overhang.

The radial cutting force (F_r) pushes a mill away from a workpiece. In peripheral (slab) milling, this force causes bending of the arbor that carries the mill. On the other hand, in face and end milling, the bending force (F_b) is the combined effect of the tangential force (F_t) and the radial force (F_r) (Fig. 30). These bending forces serve as the initial data for the bending analysis of either the arbor or the mill body, depending on the case.

Fig. 30. Cutting forces in face milling.

The bending force (F_b) can be decomposed into two components in another manner, with one of the components acting in the direction of the feed motion. This particular component is essential for the engineering analysis of a machine's feed drive.

There are various methods available for calculating cutting forces. Some rely on empirical equations that incorporate parameters such as depth of cut, width of cut, mill diameter, and coefficients and power exponents, which are selected based on different factors. Alternatively, simpler methods involve calculating the tangential force (F_t) using a specific cutting force (k_{c1}) and the average chip thickness (hm).

The specific cutting force (k_{c1}) is a force determined through experimentation, representing the force required to remove a chip area of 1 mm^2 (.0016 in^2) with a

W thickness of 1 mm (.04 in). To determine the actual specific cutting force (k_c) for the undeformed material chip with an area of 1 mm^2 (.0016 in^2) and an average thickness of hm, the following equation is applied:

$$k_c=k_{c1}\times hm^{-mc} \quad \text{(M8)}$$

The material factor mc represents experimental data that along with the specific cutting force (k_{c1}), reflects the machinability properties of a milled material.

A milling cutter removes the material layer that has undeformed cross-section area (A). If this area features the depth (a) and the width (b), the tangential cutting force (F_t) is described by the equation (M9) below:

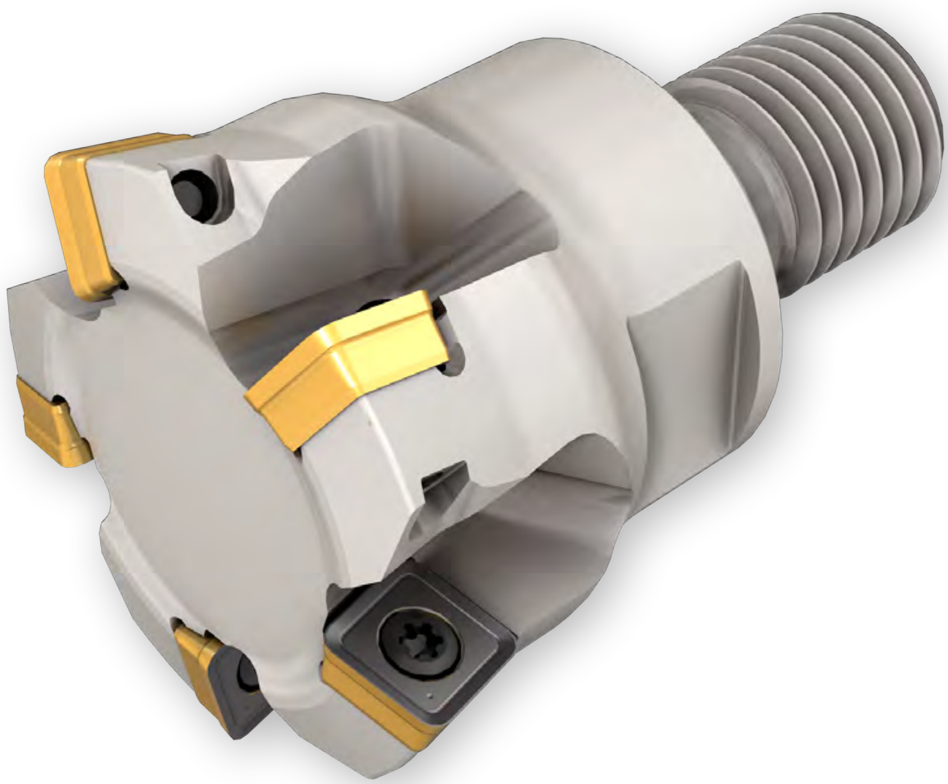
$$F_t=A\times k_c=a\times b\times k_{c1}\times hm^{-mc} \quad \text{(M9)}$$

Once the tangential cutting force (F_t) is found, the other components of the total cutting force (F) - the radial cutting force (F_r) and the axial cutting force (F_a) - can be determined using a ratio that approximates the relationship between these forces:

$$F_t:F_f:F_r=1:x:y \quad \text{(M10)}$$

Milling Thrust

In machining and particularly in milling, the term “thrust force,” or simply “thrust,” is sometimes used in technical data sources and in shoptalk to refer to the “axial cutting force.”



The coefficients x and y are derived from the empirical data that depend on the milling method and the type of material being machined.

If we know the tangential cutting force (F_t), the cutting power (Pc) can be calculated using the following equation:

$Pc=F_t \times v_f=A \times k_c \times v_f$ **(M11)**

Here v_f represents the feed speed.

By substituting the appropriate values and performing unit conversions, equation (M11) is expressed as follows:

- In the metric system, where a, b and d are in mm, k_c is in N/mm^2, and v_f is in mm/min,

$Pc=(a \times b \times k_c \times v_f)/(6 \times 10^7)$ kW **(M11a)**

- In the U.S. customary system, where a, b and d are in inches, k_c is in psi, and v_f is in ipm,

$Pc=(a \times b \times k_c \times v_f)/396$ hp **(M11b)**

On the other hand, power and torque are connected by the following interrelation:

$Mc=Pc/\Omega$ **(M12)**

Here Ω represents the rotation frequency.

Therefore, the cutting torque (Mc) can be expressed through the cutting power (Pc) in the following manner:

- In the metric system

$Mc=Pc \times 30 \times 10^3/(\pi \times n)$ Nm
$\approx 9550 \times Pc/n$ Nm **(M12a),**

where the cutting power (Pc) in kW, and the rotational velocity (n) of the milling cutter in rpm.

- In the U.S. customary system

$Mc=Pc \times 198 \times 10^3/(\pi \times n)$ lbf×in
$\approx 63025 \times Pc/n$ lbf×in **(M12b),**

where the cutting power (Pc) in hp, and the rotational velocity (n) of the milling cutter in rpm.

Example. An **ISCAR** milling cutter with six-flute exchangeable **MULTI-MASTER** solid carbide head, 25 mm (1.00") in diameter, is being used to machine a square shoulder in a workpiece made from titanium alloy Ti-6Al-4V in a single pass. The shoulder dimensions are 5 mm × 10 mm (.20" × .40"). According to **ISCAR**'s recommendations, the machining parameters are as follows:

- Cutting speed 45 m/min (147 sfm),
- Maximum chip thickness 0.076 mm (.003").

The specific cutting force k_{c1} for the mentioned titanium alloy is 1245 N/mm^2 (180.57 ksi), and the material factor mc is 0.24.

Find the cutting power and the cutting torque.

In metric units:

From the equation (M2a)
$n=1000 \times v_c/(\pi \times d)=1000 \times 45/(\pi \times 25)=573$ (rpm)

From the equation (M5)
$v_f=f_z \times z \times n=0.076 \times 6 \times 573=261.3$ (mm/min)

From Fig. 27,
$hmax=f_z \times sinAE$. $AE=arccos((r-a_e)/r)$,
where r- the cutter radius (12.5 mm)

$AE=arccos((r-a_e)/r)arccos((12.5-5)/12.5) = 53°$

$f_z=hmax/sinAE=0.076/$
$sin53°=0.095$ (mm/tooth)

Assume the average chip thickness (hm) is half of the maximum chip thickness (hmax) i.e. hm=0.038 mm.

From the equation (M8)
$k_c=k_{c1} \times h_m^{-mc}=1245 \times 0.038^{-0.24}=2729$(N/mm^2)

From the equation (M11a)
$Pc=(a \times b \times k_c \times v_f)/(6 \times 10^7)=(10 \times 5 \times 2729 \times 261.3)/$
$(6 \times 10^7)=0.59$ (kW)

From the equation (M12a)
$Mc=Pc\times30\times10^3/(\pi\times n)=0.59\times30\times10^3/(\pi\times573)=9.83$ (Nm)

In U.S. customary units:

From the equation (M2b)
$n=12\times v_c/(\pi\times d)=12\times147/(\pi\times1.00)=561$ (rpm)

From the equation (M5)
$v_f=f_z\times z\times n=.003\times6\times561=10.1$ (ipm)

From Fig. 27,
$h_{max}=f_z\times \sin AE$. $AE=\arccos((r-a_e)/r)$,
where r - the cutter radius (.50”)

$AE=\arccos((r-a_e)/r)\arccos((.50-.20)/.50)=53°$

$f_z=h_{max}/\sin AE=.003/\sin53°=.0037$ (ipt)

Assume the average chip thickness (hm) is half of the maximum chip thickness (hmax) i.e. hm=.0015”

Due to the material factor (mc) reflecting the empirical dependence found for metric units, the appropriate correction coefficient should be added to account for unit conversion.

From the equation (M8) $k_c=k_{c1}\times h_m^{-mc}=k_{c1}\times(hm\times25.4)^{-mc}$
$=180.57\times(.0015\times25.4)^{-0.24}=395.6$ (ksi)

From the equation (M11b)
$Pc=(a\times b\times k_c\times v_f)/396=(.40\times.20\times395.6\times10.1)/396=0.81$ (hp)

From the equation (M12b)
$Mc=Pc\times198\times10^3/(\pi\times n)=0.81\times198\times10^3/(\pi\times561)=91$ (lbf×in)

The required machine power consumption (P) can be estimated using the following equation:

$$P=Pc/\eta \qquad \textbf{(M13)}$$

Here η refers to the efficiency of a milling machine tool. The average efficiency of milling machining centers can be assumed as 0.9.

Simplified calculations based on equations (M8)-(M13) provide sufficient results for estimating cutting forces, cutting power, and cutting torque in machining practice. However, for more accurate analysis, it is necessary to consider factors such as cutting geometry of a mill, the engagement parameters, and other relevant aspects. Modern computer-aided engineering (CAE) systems facilitate this process.

Machining Power Calculator

The “Milling” option of **ISCAR**’s Machining Power Calculator, an advanced software for power and force analysis, enables online estimation of cutting forces, cutting power, cutting torque, and other important characteristics of a milling operation. It also provides information on the bending load on a machine spindle, power-time variation, tooth tangential force-time variation, and other relevant parameters.

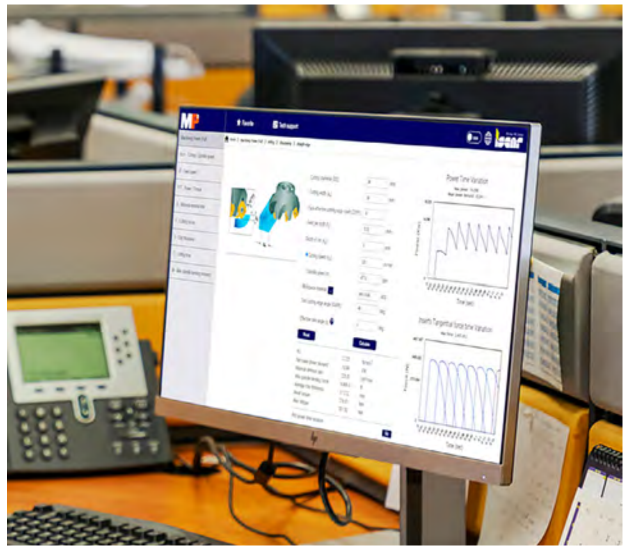


Fig. 31. ISCAR’s Machining Power Calculator is an effective tool for accurate online calculating of cutting power and cutting forces in milling applications.

Let’s now discuss up (conventional) and down (climb) milling as illustrated in Fig. 10 and Fig. 12 again. According to Newton’s third law, every action is met with an equal and opposite reaction. Therefore, the milling cutter exerts a force on the workpiece being machined, which is equal in magnitude but opposite in direction to the total cutting force. The same principle applies to the components of the total cutting force.

Preferrable Milling Method

Generally, compared to up (conventional) milling, down (climb) milling offers several advantages, including increased tool life and more evenly distributed forces. Therefore, down milling should be used whenever possible.

Let's consider a scenario where a mill with a horizontal axis of revolution is used to cut a workpiece mounted on a machine table. The force diagrams depicting the forces acting on the workpiece during up and down milling are shown in Fig. 32 and Fig. 33, respectively. It is evident that in up milling, the forces tend to lift and move the workpiece upwards, away from the table. Conversely, in down milling, the forces push the workpiece down towards the table.

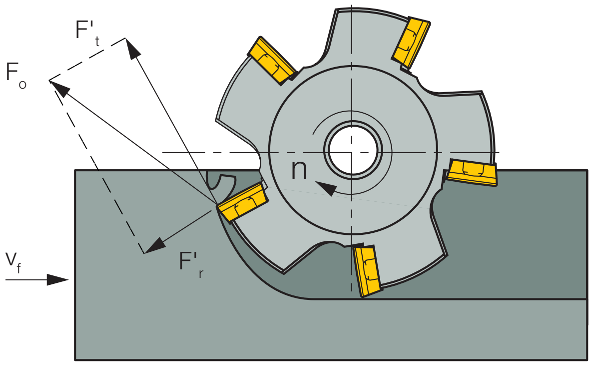


Fig. 32. Force diagram in up (conventional) milling.

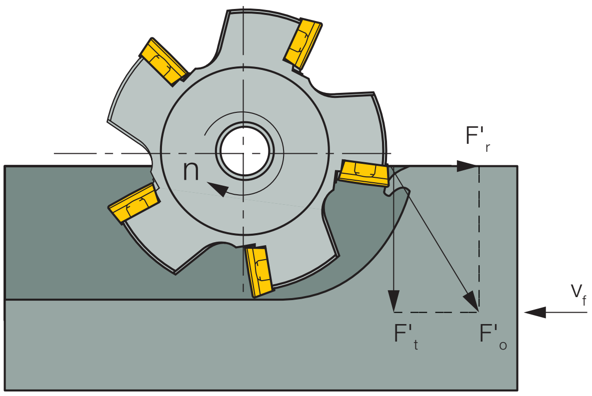


Fig. 33. Force diagram in down (climb) milling

Self-evaluation quiz

12- Do feed per tooth and feed rate mean the same?

a. Yes, these terms are the equivalent.

b. No, these terms are different.

13- The effective diameter of a profile milling cutter is

a. The nominal diameter of the cutter.

b. The largest diameter of the cutter.

c. The largest true cutting diameter of the cutter.

14- Is the maximum chip thickness always equal to the feed per tooth?

a. Yes, this is correct.

b. No, this is not correct.

15- In milling, the necessary power consumption and cutting torque are calculated based on

a. The tangential cutting force.

b. The total (resultant) cutting force.

c. The bending force.

Vibrations in Milling

Vibrations in machining, particularly in milling, are generally an unavoidable aspect of the metal cutting process. They can be either forced or self-excited and always occur alongside the cutting action. These machining vibrations are commonly referred to as "chatter," due to their specific nature that occurs whenever chips are formed. Even in cases where cutting is considered stable, it does not mean that vibrations are absent. Instead, the vibrations remain at a level that still allows for the required machining results and are considered a "no vibration" operation. In milling, vibrations are inherently connected to the nature of the process itself. The rotating mill's teeth periodically enter and exit the machined material, resulting in cyclic loads and mechanical impacts. Therefore, the cutting force fluctuates between minimum and maximum values or vice versa (Fig. 34). This dynamic response of the entire machining system leads to vibrations. Additionally, milling with high tool overhang, which significantly reduces dynamic stability, worsens the situation.

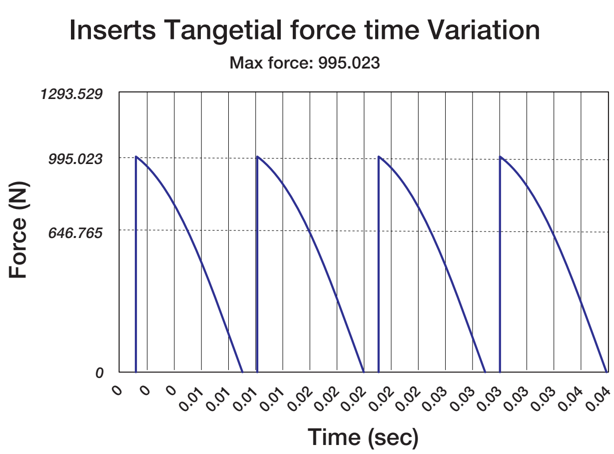


Fig. 34. Graph depicting the variation of tangential cutting force during milling a square shoulder

Figure 34 demonstrates tangential force time variation in a steel workpiece with a 25 mm diameter four-tooth indexable milling cutter (the data was obtained using **ISCAR**'s Machining Power Calculator).

In fact, vibrations in milling have a detrimental effect on performance, reducing tool life and degrading surface finish. Manufacturers strive to minimize vibrations and ideally bring them to a level that does not affect machining results. Chatter has been the subject of extensive research, which has yielded methods to model vibrations in machining. Despite their complexity, these models can be highly effective in finding ways to reduce chatter. However, such modeling requires time and various input data, including additional measurements in some cases. When faced with vibrations during machining, manufacturers typically have limited options for real-time response to reduce chatter.

Chatter Marks

In milling, chatter is characterized by a loud and discordant noise, resulting in curvy and non-uniform eye marks on the machined surface. These marks, commonly referred to as "chatter marks," can significantly degrade the surface quality.

The most common practice is to adjust cutting speed and feed, but this often results in reduced productivity. Therefore, any effective method of diminishing vibrations without compromising machining productivity would be highly desirable to manufacturers.

Essentially, reducing vibration in machining requires consideration of a manufacturing unit as a system comprising the following interrelated elements: a machine, a workpiece, a workholding device, and a cutting tool. While the influence of each element on total vibration reduction is different, improving a vibration characteristic of one element may have a significant impact on the system's overall dynamic behavior. Most efforts to protect against vibrations focus on developing more rigid machines with intelligent sensors and computer control, and advanced vibration-damping tooling.

For example, **ISCAR**'s anti-vibration shanks for mounting exchangeable **MULTI-MASTER** heads incorporate the concept of an internal absorber, ensuring quiet and safe milling (Fig. 35). The question arises: can a milling cutter, being the smallest and arguably simplest component of the system, have a dramatic impact on vibration reduction? Although producers may not have high expectations regarding the role of cutting tools in decreasing chatter, in certain cases a properly selected milling cutter can effectively eliminate vibration without adversely affecting productivity.

The right tool geometry makes cutting action smooth and stable. The geometry strongly influences cutting force fluctuations, chip evacuation and other factors, which are connected directly with vibration modes. The milling practice shows that the cutting geometry can considerably strengthen vibration damping of a tool.

Fig. 35. The anti-vibration shank for mounting exchangeable MULTI-MASTER heads incorporates the concept of an internal absorber.

ISCAR's various indexable inserts, exchangeable heads, and solid carbide tools feature chip-splitting cutting edges. Such an edge may be serrated or have chip-splitting grooves. The chip splitting action causes a wide chip to be divided into small segments, resulting in better dynamic behavior of a tool during machining, and vibration is stabilized. In rough machining, milling cutters remove a large material stock and work in heavy conditions. Significant cutting forces acting cyclically generate vibration problems. When using chip-splitting indexable inserts, it is possible to tackle these difficulties. Mills with round inserts, a real workhorse in machining cavities and pockets, particularly in die and mold making, are often operated at high overhang that affects rigidity and vibration resistance of a tool. Problems with cutting stability occur when the overhang already exceeds 3 tool diameters. Applying serrated round inserts with a chip-splitting effect redresses this situation and substantially improves robustness (Fig. 36).

In solid endmills and exchangeable heads, a skillfully defined tooth pitch is an effective way of taking the dynamic behavior of a cutting tool to the next level. For example, **ISCAR**'s **CHATTERFREE** family of solid carbide endmills (SCEM) was designed on the basis of a pitch control method. The family features an unequal angle pitch in combination with a variable helix angle. This concept ensures vibration-free milling in a broad range of applications.

Fig. 36. Chip-splitting cutting geometry can greatly contribute to the stability of milling operations in unfavorable conditions.

An assembled milling tool comprises a body with mounted cutting elements such as indexable inserts or exchangeable heads. Choosing the right body material presents an additional option for forming a chatter-free tool structure. Most mill bodies are made from high-quality tool steel grades, for which the material stress-strain behavior is similar. However, in some cases tool design engineers have identified successful material alternatives to improve vibration strength.

The **MULTI-MASTER**, an **ISCAR** family of rotating tools with exchangeable heads, provides a range of tool bodies, referred to as shanks, produced from steel, tungsten carbide or heavy metal. A steel shank is the most versatile. Tungsten carbide with its substantial Young's modulus provides an extremely rigid design, so carbide shanks are used mainly when milling at high overhang and machining internal circumferential grooves. Heavy metal, an alloy containing around 90% tungsten, is characterized by its vibration-absorbing properties, and heavy metal shanks are most advantageous for light to medium cutting operations in unstable conditions.

Typically, indexable mills that are used in long-reach applications feature an assembled design, which comprises a regular-sized milling cutter and a toolholder that mounts the cutter. This concept has important advantages such as versatility, optimized tool configuration, and effective customization. If the cutter is damaged, it can be replaced easily, while other assembly elements remain in their working state. In this case, reducing the weight of the cutter body contributes to better dynamic stability. However, weight reduction should not impair the strength characteristics of the tool.

Steel is the traditional material from which tools are made. Titanium can be used to replace steel and possesses lower density yet is characterized by high strength. Titanium also features excellent corrosion resistance with anti-wear properties being an important factor for the prolonged tool life of the cutter body. Compared to steel, titanium is characterized by poor machinability, which inevitably leads to increased manufacturing costs. A tool body design with cavities will reduce the mass of the cutter yet will increase its manufacturing costs.

Fig. 37. A shell-mill with a titanium body harnesses the advantages of additive manufacturing (AM) technology.

Modern production technologies enable solutions to overcome these obstacles. Additive manufacturing (AM) provides an effective method to achieve the complex configuration of a cutter body while minimizing machining operations. 3D printing facilitates the production of titanium made tools while assuring sustainability. Leading cutting tool manufacturers have adopted AM processes to produce milling cutters intended for machining with large overhangs. It is becoming more common to design lightweight indexable mills with titanium bodies using additive manufacturing (Fig. 37). The mill design concept also utilizes the unequal angular pitch principle to improve the vibration resistance of the cutter.

Historical Notes: Chatter

The first attempts to understand the nature of vibrations are likely associated with Frederick Winslow Taylor, a talented American engineer and researcher, renowned as a pioneer in the scientific management of labor and estimating the tool life of cutting tools. His statement, "Chatter is the most obscure and delicate of all problems faced by machinists," made at the beginning of the 20th century, remains relevant even today, emphasizing the complexity of this issue.

Cutting tool manufacturers have a limited choice of design means to reduce machining vibrations, relying mainly on tool cutting geometry and tool body material. In some cases, they may have the option of using a cutting tool with a built-in vibration-damping device. Creating a chatter-free tool with these limited resources requires considerable skill and ingenuity. However, it is feasible, and the milling solutions highlighted in the above examples affirm the possibilities.

Oscillation Cutting: Vibrations That Enhance Machining Processes

In machining, however, vibrations can not only be a harmful factor but also be beneficial. Oscillation cutting is a machining technique that combines the primary motion with the additional oscillatory motion of a cutting tool relative to a machined workpiece to break chips. This technique has primarily been utilized in turning and drilling, but it has also demonstrated successful applications in specific milling operations.

Self-evaluation quiz

16- Chatter is

a. A specific type of vibrations that characterize machining processes.

b. The type of specific forced vibrations, caused by the variation of a tangential cutting force.

c. The type of vibrations that occur when the tooth of a milling cutter periodically enters machined material and leaves it.

17- Today in shop-floor conditions, the most common practice in trying to reduce vibrations in milling is

a. Using anti-vibration tool holders.

b. Computer modelling of the process with the appropriate real-time response.

c. Changing cutting data.

18- Can the cutting geometry of a milling cutter be a design tool of anti-vibration solutions?

a. Yes.

b. No.

19- Does the material of a cutter body enable diminishing vibrations in some milling applications?

a. Yes.

b. No.

Advanced Milling Methods

The advancement in machine tools and control systems have enabled the realization of efficient machining techniques. These methods include:

- High speed machining (HSM),
- Peel milling (slicing) and trochoidal milling,
- High-efficiency machining (HEM),
- High feed milling (HFM),
- Plunge-in milling ("plunging"),
- Turn-milling.

The mentioned methods primarily aim to substantially improve milling performance, which makes them worthy of brief consideration. Understandably, they establish appropriate requirements for milling tools.

High Speed Machining (HSM)

Often high speed machining (HSM) is emphasized as "a high-efficiency method of modern machining with high spindle and feed speed". High speed machining may refer to:

- High cutting speed machining
- High spindle speed machining
- High feed speed machining

These three speeds are interrelated. Increasing spindle speed automatically results in increasing feed speed as well, and likewise higher cutting speed requires a correspondingly higher spindle speed. As cutting speed varies in direct proportion to the diameter of a rotating tool, for tools of different diameters, different spindle speeds are required to ensure that the cutting speed is identical.

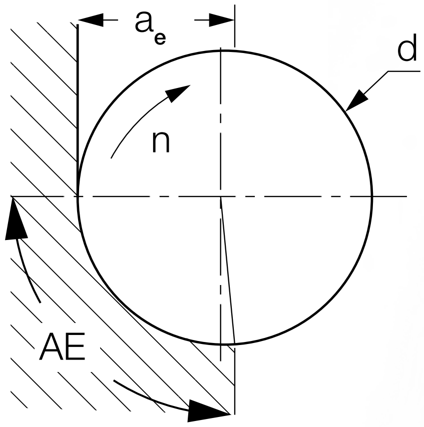
A cutting speed is also a function of several factors, where a workpiece material and a cutting tool material are dominant. Depending on the cutting tool material, the recommended cutting speed for the same workpiece material may be quite different. A good example of this is machining nickel-based high temperature alloys by cemented carbide and whisker ceramic tools. At the same time, in machining aluminum, for instance, "normal" cutting speeds are significantly higher than in machining the high-temperature alloys.

Historical Notes: HSM Preliminary

In the 1920s and 1930s, German researcher Dr.-Ing. Carl Salomon conducted a series of experiments to measure cutting temperatures against corresponding cutting speeds during the machining of certain engineering materials. The results led the researcher to hypothesize that the cutting temperature rises with an increase in cutting speed until the speed reaches a specific critical value. Beyond this point, the temperature reduces despite further increases in speed. Therefore, Salomon concluded, there is a range of cutting speeds higher than usual, within which the cutting temperatures are similar to those observed in conventional cutting. According to Salomon's hypothesis, this range corresponds to speeds that are 5- 10 times higher than common values.

Today, the term "high speed machining" usually relates to high speed milling. This milling method is characterized by shallow, light cuts combined with high rotational velocity of a tool. As previously mentioned, a mill contacts a machined material by arc (Fig. 26) that is measured by the angle of engagement (AE). Decreasing this arc (i.e. width of cut a_e) diminishes the heat load on the mill's cutting edge. In addition, it increases the interval during which the edge is not involved in cutting, providing more time for edge cooling. Therefore, reducing the width of cut (a_e) allows for a higher cutting speed (v_c).

One example is an approximate plot of the cutting speed (v_c) against the width of cut (a_e) and the angle of engagement (AE) when milling a workpiece from titanium alloy (Fig. 38). In milling full slot directly from solid the width of cut is equal to the diameter of a tool (d). In comparison with this case, in milling square shoulder with a_e less than 0.1×d (with AE smaller than 37°) the cutting speed may be increased by 150-200%!'



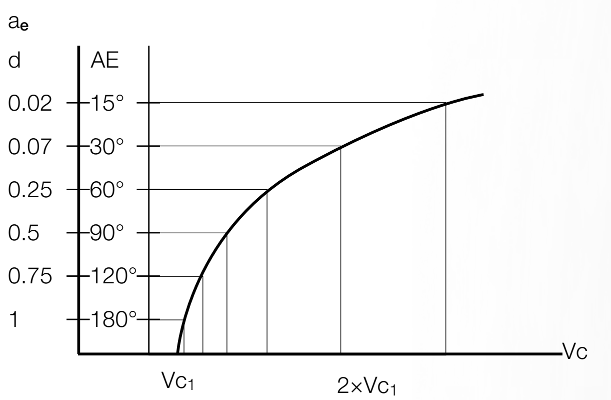


Fig. 38. Cutting speed as a function of radial engagement.

Another example is milling with a ball-nose cutter with small depth of cut - an operation that is common today for machining hard steel in die and mold making. In the case of a ball-nose cutter of 10 mm (.395”) in diameter and 0.2 mm (.008”) depth of cut, facilitating cutting speed of 120 m/min (394 sfm) requires considerable rotational velocity: 13642 rpm.

Hence, high-speed rotation, whether that be milling with small radial or axial depth of cut, is a typical attribute of high speed machining. Successful HSM relies on a key element chain comprising a machine tool, an effective machining strategy, proper toolholding, and a cutting tool. The low-power multi-axis machine tools, designed especially for HSM, feature high-torque characteristics, high-velocity drives, effective controllers and intelligent software. They are capable of realizing various machining strategies which were developed for ensuring maximum efficiency.

Today, metalworking has in its arsenal highly reliable tool holders designed for secure tool mounting in an expanded range of rotational speeds. Under such conditions the cutting tool - the element that directly contacts a machined part during a cutting operation - can be a limiting factor in maximizing the potential of advanced machine tools . This element is much smaller and less complicated compared to machine tools and holders. Each improvement in the last chain element - the cutting tool - may be crucial. The cutting tool industry is far from stagnation; the branch is on the constant move in developing new solutions to meet the demands of changing metalworking technologies.

Time has not radically changed principal tool requirements: it is expected to be more durable and more efficient when cutting at considerably increased cutting speeds and feed rates. Lowering machining allowances leads to additional tightening tool accuracy parameters. An ideal product is a precise and high-balanced tool that ensures high performance in combination with excellent tool life when cutting at high rotational velocities.

Solid endmills feature higher accuracy and better axial symmetry when compared with indexable cutters. Typically, solid endmills are less in diameter and naturally require higher rotational velocity even for the same cutting speed. This explains why the majority of HSM tools are solid. Normally, such tools are made from coated cemented carbides, although in recent times cutting ceramics as a tool material has become popular for high-speed machining of high-temperature superalloys. Nevertheless, selecting a solid milling cutter for HSM may be difficult.

Usually, the overhang-to-diameter ratio for solid carbide endmills (SCEM) is greater when compared with indexable tools. Such a feature, in combination with a flute shape that weakens a tool cross-section, demands specific attention to the vibration strength of a SCEM. To improve chatter stability, tool engineers often make a tooth angular pitch unequal, and a flute helix variable. This violates the principle of axial symmetry and may give a reverse result. Therefore, an optimal, intelligent design for solid carbide endmills requires engineer ingenuity and appropriate compromising (Fig. 39).

High-speed rotation generates centrifugal forces. In HSM, when compared with traditional machining methods, these forces grow exponentially and turn into a significant load on a cutting tool which determines the tool's durability. In indexable milling, high centrifugal forces may cause insert clamping screws to break, inserts to loosen and a cutter body to fail. Formed fragments can not only damage a machine and a machined part but can be very dangerous to the operator. In such conditions, cutting tool manufacturers are compelled to consider the design and technological means necessary to ensure appropriate reliability of their products. Hence, the focus on indexable milling cutters should consider secure insert mounting and a robust body structure.

Fig. 38. ISCAR's 7 flute solid carbide endmill, intended for HSM, features different helix angles, variable pitch, and chip-splitting grooves on the cutting edges.

It is known that the ambition of a tool design engineer is to make an indexable cutter body, and in particular, an insert pocket surface, as hard as possible in order to increase wear resistance. However, the higher the hardness, the faster the body of a rapidly rotating tool breaks down. Hence, finding an optimal equilibrium between strength and wear is another important task in searching for effective HSM tool solutions.

To reduce centrifugal forces, a cutter body ought to be axially symmetric and highly balanced. There are international and national standards and norms that specify tool balancing grades. When designing indexable milling tools intended for HSM, it is very important to ensure the mass distribution of the body is symmetrical with the body axis. As this theoretical balance relates to a virtual object, it cannot replace the physical balancing of a real body if needed but can substantially diminish the mass unbalance of a future product making the "physical" balance much easier. However, having highly engineered a balanced vibration-proof tool is half the battle. In HSM, the dynamic characteristics of the tool cannot be separated from a toolholder. For example, balancing the tool should be done in assembly with the toolholder – this is a single way to fulfill requirements of accuracy, reliability, and safety.

The metalworking industry adopted high speed machining in the 1990s. HSM that features small stocks per pass, has distinct advantages such as lower power consumption, less heat generation, and better surface finish. This method was engrained in various industrial branches and caused serious changes in technology and machine tool engineering. In particular, HSM is now widely used in the machining of aluminum in aerospace and automotive industries, as well as steel, especially hard steel, in die mold making. Accurate high speed machining, which features low stock removal, is a logical extension of producing workpieces by modern methods such as precise casting, metal injection molding, and 3D printing.

To conclude, high-speed machining has influenced the need for specific requirements of a cutting tool and toolholder. By meeting these demands, HSM has become a trusted highly engineered, high-speed spindle operation with maximum efficiency. Moreover, high speed machining has given rise to several derivative subtype methods, which will be discussed in the sections that follow.

Peel Milling (Slicing) and Trochoidal Milling

The introduction of machine tools with significantly increased rotary and linear velocities was the success to efficient high-speed machining (HSM) methods. Peel milling, also known as slicing, was one of these methods. The main principle of peel milling is its high depth of cut (usually, no more than five-tool diameters) when coupled with a low width of cut (typically, up to 0.2 of a tool diameter). This combination features significant advantages.

Decreasing the width of cut, as previously detailed, reduces heat load on a cutting edge and allows increasing cutting speed. In peel milling, the cutting speed can be higher when compared with traditional milling methods. The low width of cut significantly diminishes the radial component of a cutting force, which causes mill bending and vibrations. This ensures high operational stability and facilitates an increased depth of cut.

Radial chip thinning enables higher feeds to maintain the required accurate chip thickness. Therefore, milling with a small radial engagement and a substantial depth of cut performed at high cutting speeds and feed rates is a useful source for improving machining productivity. Moreover, such a machining method provides gradual, uniformly distributed wear along the whole cutting edge, thus increasing tool life. Peel milling has proven to be productive in milling deep shoulders and wide edges. The slicing technique is successfully applied to rest milling – a machining process where a small diameter tool cuts various hard-to-reach areas, such as cavity corners.

The advance of computer numerical control (CNC) and computer-aided manufacturing (CAM) systems have generated further improvement: trochoidal milling with a complicated tool trajectory instead of a linear feed motion - suitable for peel milling. In mathematics, a trochoid is the curve, generated by the point of a circle rolling along a guide without sliding. In trochoidal milling, a cutting tool moves along a curve slicing thin and slim material layers. Commonly, the curve is a circular arc (semicircle), and the tool returns to the initial point by the arc chord and then repeats the path with a small stepover. In this case, the tool path looks like the letter “D”. Milling along the curvilinear trajectory facilitates constant loading of a cutting edge and eliminates a sharp increase in load when entering the material.

In addition to the D-shaped path that is now considered “classical”, today, most advanced machines with high-end control systems are much more complex. Trochoidal tool trajectories minimize non-cutting time and optimize machine unit motions. Trochoidal milling is known to be very effective in machining deep slots, pockets, and cavities and is also a very promising method to mill hard and difficult-to-cut materials, in particular titanium and high-temperature superalloys (HTSA). In addition, trochoidal milling is extremely useful for improving performance when cutting in unstable conditions: non-rigid workpieces, thin-wall areas, poor work holding devices, etc. And even more so, uniform and considerably reduced tool loading makes trochoidal milling efficient and applicable for micro machining.

What are the features of a high-performance trochoidal milling cutter? To begin with, the trochoidal milling cutter must be suitable for high-speed machining. This relates to appropriate accuracy parameters, balancing, safety when operated at considerable rotational speeds, and more. Milling with high depths of cut increases the tool's overhang while the dynamical behavior of a cutter is crucial to ensure machining stability. When milling with a low width of cut, only one tooth engages the workpiece material at any given time. Optimizing a contact area along the tooth is an important factor for stable milling, and the cutter with the most favorable tool cutting edge inclination is a principal part for finding the best solution. The effective evacuation of the thin chips, which are generated when trochoidal milling, does not require a large chip gullet in the cutter.

Even a brief examination of the above shows that multi-flute solid carbide endmills (SCEM) or assembled mills carrying replaceable solid carbide heads comply with the requirements in the best way. Indeed, SCEM's represent most trochoidal milling tools today.

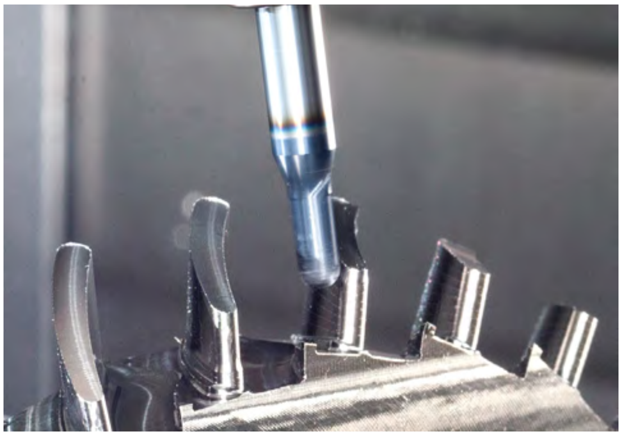


Fig. 39. Trochoidal milling is an effective method for improving productivity during the rough machining of geometrically complex surfaces.

Accurate Roughing

Often, when a part with complex profiles and slots is made from a solid material, the use of trochoidal milling can be quite effective. This technique shapes the part very close to its final form. The remaining small allowance is removed in the next stage: high speed finish milling. In the aerospace industry, the mentioned process is typically used to produce blisks (bladed discs), blings (bladed rings), and impellers (Fig. 39). Therefore, trochoidal milling may be defined by the rather oxymoronic term: "accurate roughing".

High-Efficiency Machining (HEM)

More generally, "high-efficiency" can refer to a highly-productive machining strategy with high output-input ratio. At the same time, in the context of milling techniques, high-efficiency machining (HEM) is a milling method much like high-speed machining (HSM), which utilizes a large axial depth of cut and a small radial depth of cut in combination with high rotational velocity (spindle speed) of the tool. However, the radial depth of cut varies depending on the angle of tool engagement to facilitate constant chip thickness per cutting edge during tool rotation. This method assures the efficient usage of a milling tool for the uniform development of wear that covers a large section of the tool's cutting edge. HEM is often referred to as "dynamic milling" and features productive rough milling operations.

High-efficiency machining (HEM) offers two major advantages, in addition to its high metal removal rate. Firstly, it diminishes the impulse load on the technological system, which includes a machine tool, a milling cutter, and a workholding fixture. Secondly, it reduces tool wear development, thereby extending the tool's lifespan.

However, HEM demands the right capabilities from both CAM and CNC systems to generate the necessary toolpath. The control system of a machine tool must have a high data processing ability, and the machine should be suitable for working in conditions with high fluctuations in feed speed. Therefore, dynamic milling is not a universal machining strategy that can be used on every milling machine.

The requirements for an HEM tool are the same as those for a cutter intended for trochoidal milling.

High Feed Milling (HFM)

Highly-productive machining with large-sized milling cutters can be likened to the work of a heavy excavator digging sand with a big bucket. The full sand bucket, operated by a powerful engine, slowly moves a large volume of waste material. At the same time, there is an alternative method for efficient excavating. Imagine a more compact track trencher with a rapidly moving digging chain. Each link of the chain removes a small volume of sand but does it fast. In metal cutting, this trencher is a high feed mill, which machines at shallow depths of cut (a_p) but with a feed per tooth (f_z) that is far higher than the usual rates - millimeters as opposed to tenths of millimeters.

The method of rough machining with significantly increased feed per tooth – known as high feed milling (HFM) or fast feed (FF) milling – found its industrial application in the 1990's. Die and mold making was one of the first industries to adopt HFM into its production practices, following a massive increase in customer demands for reduced die and mold manufacturing time. High feed milling answered this need while providing an effective tool for boosting productivity. The HFM method is based primarily on two principles: the cutting geometry of a milling tool and the high-speed feed drive of a machine.

It is important to remember that the maximum chip thickness (h_{max}) is a function of the tool cutting edge angle (κ). For the same feed per tooth, decreasing this angle results in thinner chips. Consequently, to maintain the necessary maximum chip thickness, an appropriate increase in feed per tooth is required (Fig. 40).

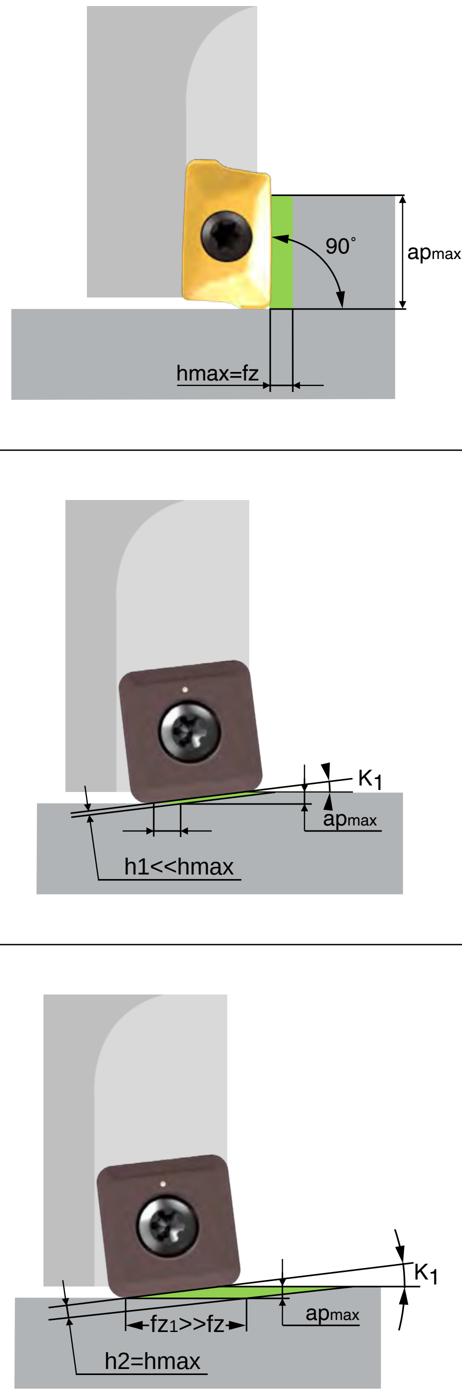


Fig. 40. Geometrical relations between feed per tooth, cutting edge angle, and chip thickness.

A typical high feed milling tool features a small cutting edge angle, normally 9-17°. This design characteristic results in three important outcomes. The first is the possibility of considerably increasing feed per tooth due to a chip thinning effect. For example, in face milling low-alloy steel, 0.2 mm/tooth (.008 ipt) is a near maximum value feed, but high feed milling the same material with a 2 mm/tooth (.08 ipt) feed is normal. The second is a shallow depth of cut (a_p) that ensures this geometry for the tool. Milling with reduced DOC diminishes cutting force and power consumption. And the third point relates to minimizing the radial component of the cutting force combined with maximizing its axial component, which acts toward the axis of the machine tool spindle, i.e. the direction of the maximum machine tool rigidity. This improves machining stability.

Increasing feed per tooth means greater feed speed that requires the appropriate feed drive of the machine tool. In the above example of high feed milling low-alloy steel, the feed speed may be 7000-9000 mm/min (275-355 ipm) – the next higher order versus conventional values.

Recognizing market needs, machine tool manufacturers developed a variety of machines intended specifically for high feed milling. These relatively low-power machines have “triple high” characteristics: high-torque, high-thrust spindle, and high-speed feed drive. The machines feature advanced computer numerical control (CNC) hardware and software. Introducing HFM substantially changed the concept of rough milling. Instead of intensive material removal at large depths and width of cut by using high-power machines, the method proposed extremely productive milling at shallow depths by low-power machines fitted with a cutting tool that runs very fast.

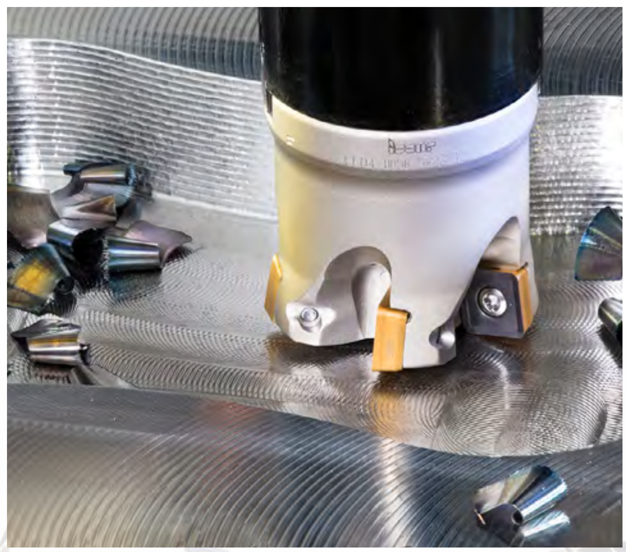


Fig. 41. A versatile indexable high feed milling tool is suitable for productive rough machining open planes, 3D surfaces, pockets, and cavities.

The fast feed milling method has since undergone some interesting changes. Originally considered as an effective way for rough machining cavities and pockets that was typical for die and mold making, HFM soon proved advantageous in face milling ("fast feed facing" or "triple F"). The diameter range of the high feed milling cutters was increased, and the group of engineering materials suitable for cutting by the HFM method was expanded. Fast feed milling quickly penetrated many industrial branches.

Radius for Programming

In CNC programming, a fast feed cutter is often specified as a 90°mill with a corner radius. This imaginary radius, referred to as the "radius for programming", is an important data because it defines the maximal thickness of a cusp (scallop) and deviations from the theoretical profile of a surface that is generated by such a specification.

It began to be more than an effective technique for the applicative niche of die and mold making, embracing all metal cutting areas as a generally recognized productive method (Fig. 41). Steel and cast iron may be known as the main "consumers" of fast feed milling, but stainless steel, titanium, and even high temperature superalloys can be successfully machined by the HFM method as well (and it is already not uncommon today). This in turn led tool manufacturers to introduce a variety of fast feed milling cutters in different forms. Indexable or solid in concept, they can have shank or arbor type design configurations, integral or modular body structures, and cutting geometry that varies according to the machined material group.

Plunge Milling ("plunging")

Plunge milling, also known as "plunge-in milling" and "plunging," is a method of rough milling during which a tool moves directly downward into a workpiece, with feed directed along the tool axis. A typical plunge milling cycle operates as follows: the tool "plunges" into the workpiece, cutting the material with its end rather than its periphery, then the tool lifts out, moves in a linear stepwise manner, and repeats the cycle. Plunge milling (Fig. 42) generates a serrated surface with cusps that can be reduced by decreasing linear stepover; however, the surface typically requires additional milling for a better finish. In shoptalk, the metalworking industry jargon, a plunge milling tool is often referred to as a "plunger."

In plunge milling, the significant component of the total cutting force acts axially – exactly in the direction of the highest rigidity of a machine tool. This minimizes bending forces and allows for productive milling with excellent straightness of machined surfaces even at high tool overhang. However, the substantial axial cutting force can significantly increase the load on the machine spindle's bearings.

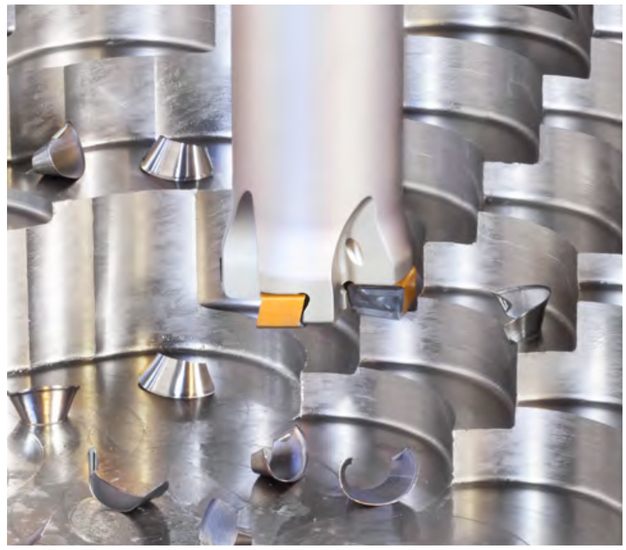


Fig. 42. Plunge-in milling enables productive rough machining to pre-shape complex 3D surfaces.

Sculpturing

In plunge milling, the term "sculpturing" typically refers to pre-shaping of 3D surfaces and deep recesses. More broadly, in metal cutting, this term can also encompass methods such as engraving and chiseling, among others, which are used to form surfaces.

The advantages of plunge milling are as follows.

1- Plunge milling offers a higher metal removal rate compared to traditional milling methods.

2- It provides increased machining stability, which results in reducing vibration and noise, especially when milling at high tool overhang.

3- This method is highly effective for rough milling of deep cavities, pockets, slots, and walls

4- Plunge milling produces small, manageable chips that facilitate easy chip evacuation.

At the same time, plunge milling has several disadvantages, including:

1- Limited applications - this method is primarily used for rough machining, particularly deep slots and cavities.
2- The high axial load places additional demands on the machine spindle's bearings.
3- CNC programming of tool trajectories, especially for complex machine surfaces, can be challenging.

High Speed Plunging

Various research efforts to increase plunge milling performance associate enhanced productivity with high speed plunging at extremely high feed speeds. This promising plunging concept resembles the fast movements of a needle in a sewing machine. Despite its potential, the concept presents challenges as it requires high-speed machines capable of handling quick-cycle changes of accelerated and decelerated movement in the direction of the machine spindle axis.

Turn-Milling

Turn-milling is a process whereby a milling cutter machines a rotating workpiece. This method combines milling and turning techniques and has many advantages, but only relatively recently the introduction of multitasking machine tools has allowed turn-milling to display its benefits.

For years, even under mass adoption of CNC technology, development of metal cutting machine tools was traditional enough, when progress of specific machines like turning, milling or drilling moved towards a separate direction. If machining centers already successfully integrate machining by rotating tools – milling and drilling – turning CNC machines continued with their own progress. Looking for new ways to make manufacturing process more efficient by reducing setups of a machined part and its transfer from one machine to another led to adding a tool head with rotary drive to typical CNC turning machines and allowed realization of turn-milling.

Today modern multitasking machine tools feature additional axes of the head movement, advanced control systems and upgraded software that provide the opportunity to perform the majority of machining operations with only one setup per workpiece.

In turn-milling, there are two principal kinds of machining: peripheral (Fig. 43a), when axes of a workpiece and a cutter are parallel; and face (Fig. 43b), for which these axes cross. Peripheral turn-milling is similar to milling by helical interpolation and may apply both to external and internal surfaces of the revolution, while with the use of face turn-milling only the external surfaces can be machined.

Despite the fact that turn-milling seems to be very similar to turning ("turning by rotating mill"), there is a substantial difference between these two machining processes. The cutting speed in turn-milling is defined by the peripheral speed of the milling cutter and not by the rotary velocity of the workpiece like in turning. The workpiece rotation relates to feed. Usually, when discussing turn-milling, it typically refers to face turn-milling, which is the more common method.

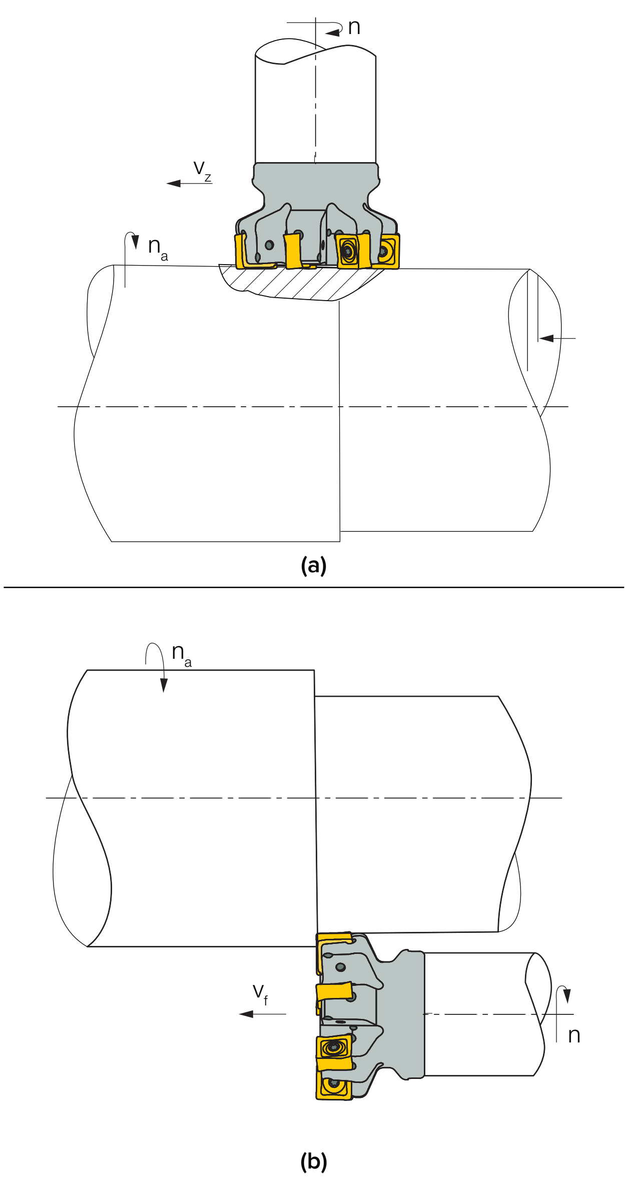
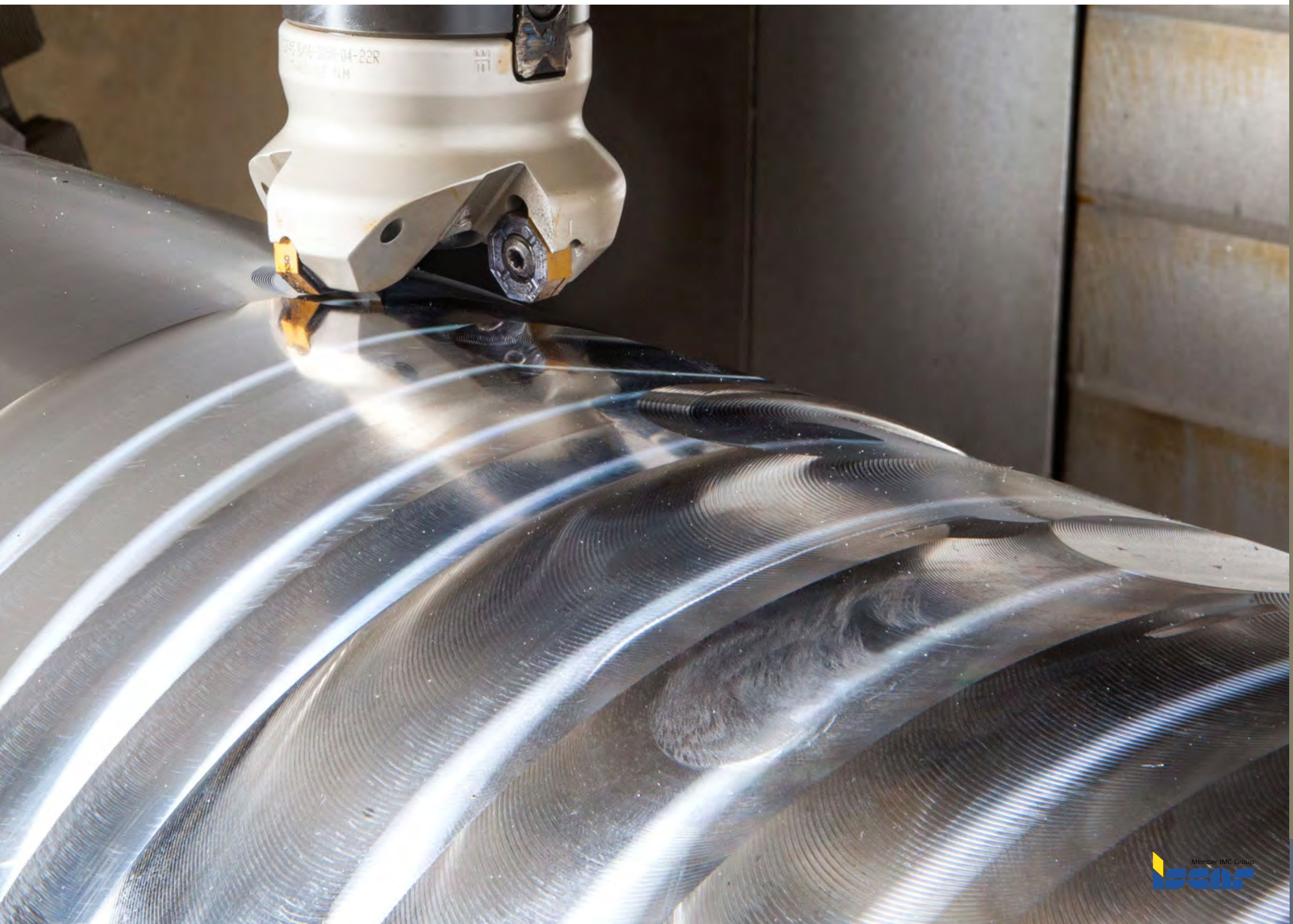


Fig. 43. Peripheral (a) and face (b) turn-milling operations.

What are the advantages of face turn-milling and where is its application practical?

- First of all, machining of non-continuous surfaces may cause interrupted cutting (various grooves, undercuts, etc.). In classical turning, this operation results in unwanted impact load, poor surface finish and early tool wear. In contrast to turn-milling, the tool is a milling cutter that is intended exactly for interrupted cuts with cyclic load.
- Machining materials produces long chips. In turning, chip disposal is difficult; and finding a proper chipbreaking geometry of a cutting tool is not such a simple task. The milling cutter used in turn-milling generates a short chip that considerably improves swarf handling.
- Take for example, machining eccentric areas of rotating components such as crankshafts or camshafts: in turning, off-center masses of these components (crank journal, eccentric cam, etc.) cause unbalanced forces that adversely affect performance. Turn-milling with its low rotary velocity of a workpiece gives the possibility to prevent this negative effect.
- Also, consider machining heavy-weight parts (Fig. 44): Their rotation, which defines cutting speed in turning, is connected with limitations of the main drive of a machine tool. If the drive does not allow rotation of large masses with required velocity, cutting speed is far from the optimal range; and turning performance will be low. Turn-milling provides a way to overcome the above difficulties effectively.



However, productive machining with the use of the turn-milling method demands right cutter positioning with respect to the workpiece, correct choice of insert geometry and tool path. Cutter positioning, for instance, influences form errors, and insert geometry – surface finish. The questions of applying turn-milling, tool choice and defining cutting data deserve fuller consideration and should be examined specifically.

Introducing turn-milling into the manufacturing process can solve serious problems and substantially improve a metal shop's output. Nonetheless, improving productivity using this promising method assumes that a modern machine and properly selected cutting tools are available.

Fig. 44. Turn-milling is an effective method in rough machining heavy rotary parts.

Self-evaluation quiz

20- High speed machining (HSM) is characterized by

a. Shallow cuts combined with high feed per tooth.

b. Shallow cuts and significant width of cut combined with high rotational velocity of a mill.

c. High-speed rotation of a mill combined with shallow cuts.

d. High rotational velocity of a mill combined with a feed motion along the mill axis.

21- Trochoidal milling is a method of

a. High speed machining.

b. High feed milling.

c. Plunge milling.

d. Turn-milling.

22- Dynamic milling is suitable for every high-speed machine tool.

a. Correct.

b. Not correct.

23- The high feed milling concept utilizes chip thinning effect.

a. Correct.

b. Not correct.

24- High feed milling cutters features

a. Small cutting edge angles.

b. Small lead angles.

25- Plunge milling is suitable for

a. Roughing.

b. Finishing.

c. Both roughing and finishing.

26- Face turn-milling is not suitable for machining non-continuous surfaces.

a. Correct.

b. Not correct.

ISCAR's Milling Line

ISCAR's milling line is richly diverse and it features a great variety of products. These products are divided into different tool families based on their design concept, such as indexable milling cutters (Fig. 45), solid endmills (Fig. 46), and assembled tools with interchangeable milling heads (Fig. 47 and Fig. 48). Each family includes various types of tools to cover a wide range of applications, including milling plane faces, complex surfaces, slots and grooves, threads, teeth of gears and splines. The tools are intended for different milling types: roughing, semi finishing, and finishing of all engineering material groups. Some tools are specifically designed for high speed milling (HSM), high feed milling (HFM), plunge milling, and other techniques. **ISCAR** produces milling tools from all types of cutting materials, including high-speed steel (HSS), tungsten carbides, ceramics, polycrystalline diamond (PCD), and cubic boron nitride (CBN). The most common cutting material is coated tungsten carbide.

Table 4 characterizes a typical diameter range of **ISCAR**'s standard program of milling tools.

Table 4. Diameter range for typical ISCAR's standard milling tools*

Tool Design Concept		Tool Type	Nominal Diameter Range	
			mm	Inches
Indexable Milling Cutters		Shell mills and mills with shank	6-315	.313-12
		Disc-shape slotting and slitting cutters	21-425	.78-9.87
Solid Endmills			0.2-25	.125-1.00
Milling Tools With Exchangeable Solid Heads	MULTI-MASTER family	Endmills	5-32	.187-1.25
		Disc-shape cutters	7.7-28	.303-1.091
	T-SLOT family	Disc-shape cutters	25-50	1.00-1.50
	T-FACE family	Face mills	32-50	1.50-2.50

* This table is for general information only. For fresher data, refer to the updated **ISCAR** catalogues.

Indexable milling cutters (Fig. 45), which carry indexable inserts, encompass a broad diameter range and can carry out various milling operations. Tungsten carbide is the main material used to produce the inserts, but the inserts from ceramics, cermet, and ultra-hard materials like CBN or PCD are also common. The indexable design concept allows for rational utilization of cutting materials. The indexable milling cutters can withstand substantial cutting forces and are extremely effective in achieving high metal removal rates, particularly in rough and semi-finish operations.

On the other hand, one-piece (integral) solid endmills (Fig. 46) are produced through grinding, yielding high tool accuracy and making them perfect for precise, finish milling. However, due to economic factors, the diameter range of solid endmills is typically capped at 25 mm (1").

Assembled tools with exchangeable solid cutting heads integrate the advantages of both indexable and solid designs, resulting in a concept where the strengths of each design complement each other. These tools exist as an "in-between" solution to solid endmills and indexable cutters, ensuring both high precision and efficient use of cutting materials. Thus, they can be determined by a slightly paradoxical term: "indexable solid mills." **ISCAR**'s milling line includes three families of the tools with exchangeable solid heads: **MULTI-MASTER**, **T-SLOT**, and **T-FACE**. In **MULTI-MASTER** (Fig. 47), a cutting head is centered in a tool body (referred to as "shank" in the family) by a short taper, creating a face contact between the head and shank, and is secured in the shank with a special-profile thread. In **T-SLOT** (Fig. 48) and **T-FACE**, a head is centered by a cylindrical area and is secured by a screw, while torque is transmitted from the body to the head through a specially designed spline connection.

The success of these families has brought an additional intriguing solution: integrating heads with indexable inserts into the bodies of **MULTI-MASTER** and **T-SLOT** tools (Fig. 49). This is an excellent example of unlocking synergy through the advantageous combination of diverse design concepts.

Rough and Finish Milling

Generally speaking, everyone understands the principal difference between rough milling (roughing) and finish milling (finishing). Rough milling pre-shapes the surface of a workpiece, while finish milling forms this surface finally. Rough milling focuses on high metal removal rates, while finish milling ensures exacting accuracy and surface quality on the milled surface. As a rule, finish milling features significantly smaller machining allowances (material stock to be removed) when compared with rough milling. Therefore, when planning a milling process, rough and finish milling operations differ in terms of accuracy and surface finish requirements, machining allowances, and cutting data.

How to Determine when to Replace an Insert (Change Its Cutting Edge), a Solid Endmill or an Exchangeable Head?

The correct answers are: at the end of the tool life or upon reaching the wear limit. The tool life period or the wear limit for a cutting tool depends on various designs and operational and administrative factors. At the same time, during a machining operation, there are certain signs that can indicate the need to replace inserts, tools, or heads:

- Noticeable increase of power consumption (spindle load)
- Increased vibration and noise
- Worsening of machining accuracy and a need for frequent additional tool dimensional adjusting
- Reduced surface finish
- Occurred burrs
- A visual inspection of a cutting edge shows considerable flank wear, extensive edge chipping, cracks etc.

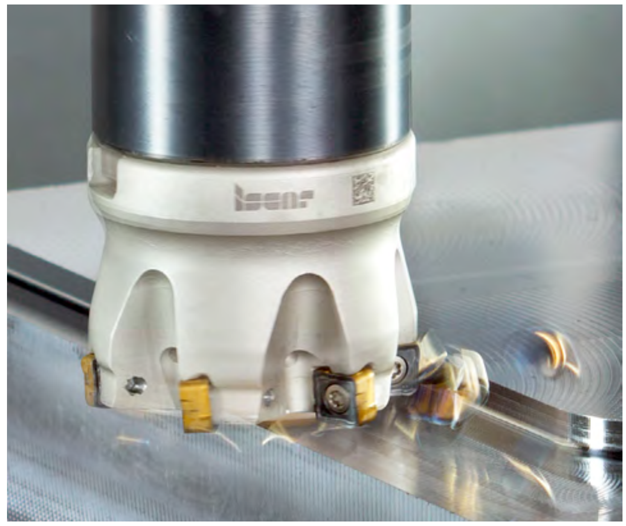


Fig. 45. Indexable milling cutters feature various insert shapes and a wide diameter range.

Fig. 46. Precise solid carbide endmills enable accurate milling of complex 3D shapes.

Fig. 47. MULTI-MASTER family provides a rainbow of exchangeable solid carbide heads.

Fig. 48. The application range of T-SLOT extends beyond milling slots and grooves. This concept has also been utilized in the milling of threads, gears, and splines. Furthermore, the T-FACE family has adopted this concept for use in face milling applications.

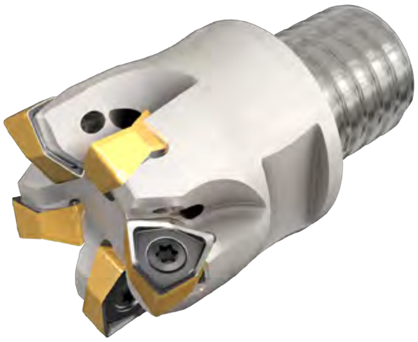


Fig. 49. The MULTI-MASTER, as well as the T-SLOT/T-FACE tools, are also suitable for mounting exchangeable heads with indexable inserts. This broadens the application range of the families and enhances the utilization of the tool bodies.

How to Find a Porcupine in a Shop Floor?

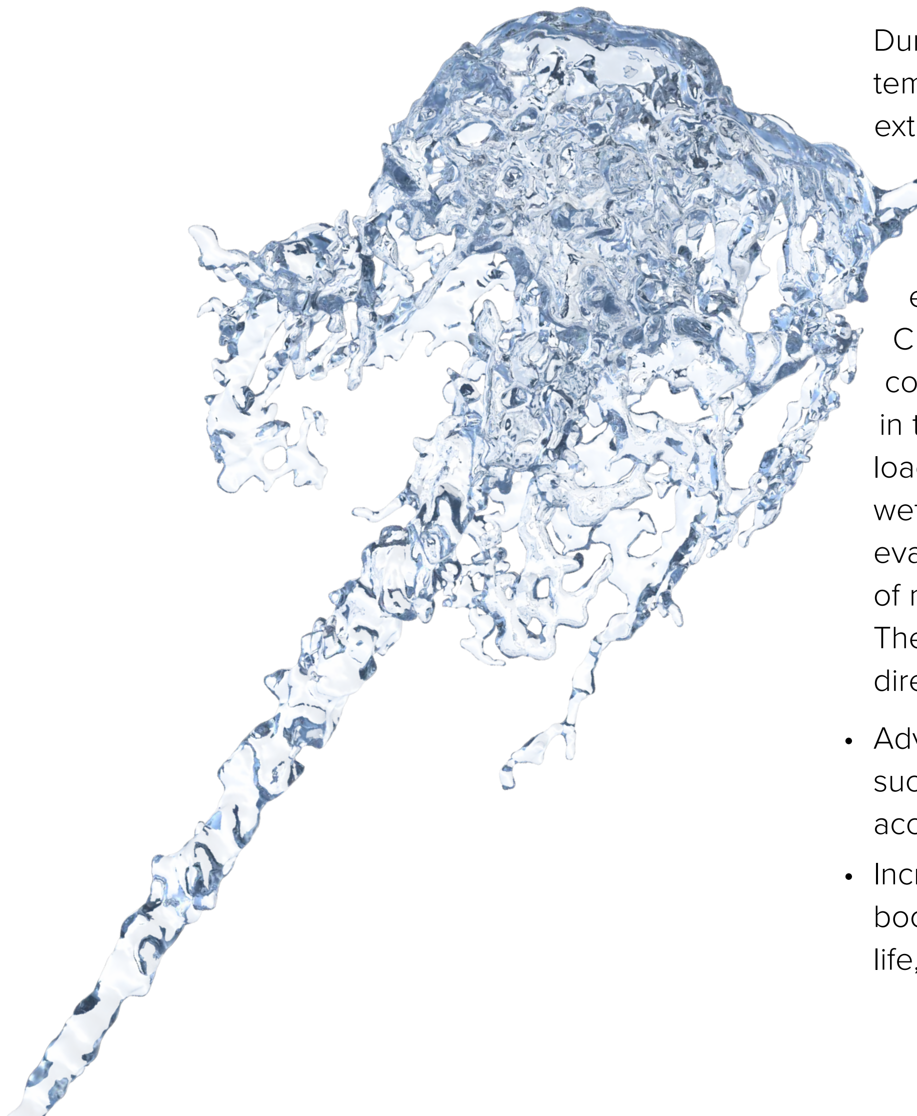
Maybe, this porky is in a barrel, and lenses will help? Indexable extended flute milling cutters (Fig. 1 on page 13) are irreplaceable tools useful for machining rough edges, deep shoulders and cavities and enabling a substantially increased metal removal rate. The tooth – the cutting blade - of an extended flute cutter consists of a set of indexable inserts that are placed gradually with a mutual offset of one another. Compared to an ordinary indexable mill whose length of cut is limited by the cutting edge of its insert, the cutting length of the extended flute cutter is significantly larger – it is "extended" due to the set of inserts. These cutters are also referred to as "long-edge cutters" and "porcupine cutters" (known as "porkies" in shop talk). In a shop floor you can find also the "barrel" – a type of profile milling cutters with large-radius cutting edges, known as segment endmills. In "barrels", barrel-shaped cutters, this radius features peripheral cutting edges, while in "lenses" – another type of the segment endmills, the arc of a large-radius circle represent the convex cutting face. Depending on the orientation of the cutting edge relative to the tool axis, segment endmills possess various configurations such as pure barrel, tapered barrel, lens, and oval or parabolic shapes. The form of the tool cutting edge determines the tool application. For example, lens-shaped tools are suitable for both five and three-axis machines, while endmills with a tapered barrel profile are intended for five-axis machines. Segment cutter designs appear in multi-flute solid endmills that deliver ultimate tool accuracy and maximize the number of teeth on the cutting tool. Machining surfaces using segment-type endmills enables a substantially increased step size compared to ball-nose cutters, thus reducing the cutting time. A three-axis CNC controlled cutting process cannot guarantee the correct position of a barrel-shaped cutting tool when machining complex surfaces. The five-axis machining concept allows taking full advantage of segment endmills.

Dry or Wet?

When milling, a major question is "Which is better: dry or wet machining?"

Throughout the world of contentious machining, the issue of – "with coolant supply" (wet) or "without coolant supply" (dry) is a common subject of discussion. To further complicate the decision, near-to-dry or minimum quantity lubrication (MQL) cutting techniques may represent a successful compromise, and therefore provide an efficient and effective answer to the troublesome question.

As in many areas of machining, making such choices is not easy, and therefore, this familiar question requires careful and informed consideration.



Wet Coolant

Wet coolant, cooling mixture, cutting lubricant, cutting fluid, and coolant are all commonplace shop-floor terms that are familiar to all involved in machining. Each expression refers to a fluid, which is used across multiple processes for both cooling and lubrication purposes.

All cutting activities generate unwelcome friction between the surfaces of the tool being used and the workpiece it is in contact with. The presence of coolant ensures that the friction between the two surfaces is reduced and by doing so makes the removal of a metal layer by the tool a great deal easier (lubrication).

During the machining process, the temperature in the cutting zone can reach extremely high levels. For example, the temperature at the point of contact between a tungsten carbide tool's cutting edge and the workpiece is estimated to be between 1100° and 1200° C (2012° to 2192° F). To mitigate this, wet coolant is applied to lower the temperature in the cutting zone and reduce the thermal load on the tool (cooling). The use of wet coolant also aids in enhancing chip evacuation and reducing the concentration of metal dust in the manufacturing area. Therefore, the supply of wet coolant is directly linked to several crucial objectives:

- Advancing process performance, such as improving machining accuracy and surface finish.
- Increasing economic efficiency by boosting productivity, enhancing tool life, and reducing tool consumption.

- Improving environmental control.

When performing an interrupted milling process, the cutting edge of the tool comes under a cyclic thermal load. Also, the ambient temperature is dramatically changed when the edge enters into, then leaves the workpiece. The tool's cutting edge is exposed to severe heat stress comparable to repeatable thermal shock. Cemented carbide, today's main tool material, is a sintered product of powder metallurgy. This material is sensitive to thermal shock load, which destroys cutting edges. When using a carbide tool, the application of a coolant supply may increase such "shock treatment" and unintentionally contribute to the failure of the tool's edge. Extreme temperatures result in plastic deformation of the cutting edge, whilst the presence of temperature differences leads to thermal cracks. This situation becomes even more exaggerated in high-heat generation milling situations, such as when machining difficult-to-cut materials or when making rough passes with significant machining allowance.

As explained, although wet cooling delivers undoubted benefits, it also has the capacity to produce several major disadvantages within the milling process. In many cases the use of a wet coolant supply is not only reasonable, but it is absolutely necessary: without wet coolant, productive milling quite often would be highly inefficient and even impossible, for example, when machining materials such as titanium and high temperature super alloys (HTSA), heat resistant steels or special-purpose alloyed hard cast iron, when friction and heat generation are considerable. Also, the flushing effect of a wet coolant supply significantly improves chip evacuation and reduces re-cutting, particularly when milling deep pockets or narrow slots.

Flood At Shop Floor

In shop talk, a cutting fluid that is supplied to a cutting zone from outside (externally) using a low-pressure jet nozzle is often referred to as "flood coolant."

Compared with traditional low-pressure wet coolant, normally delivered at 10-20 bars (145-290 psi), the relatively recent introduction of high-pressure coolant (HPC), in which the wet coolant is provided under approximately 80 bars (1160 psi) and even more (Ultra HPC), at 350 bars (5000 psi), has been a welcome development. Intensive heat generation, when using traditional low-pressure wet cooling, produces a vapor film on the surfaces of a tool and a workpiece. This layer acts as heat seal, producing an insulating barrier and making heat transfer harder, which significantly shortens tool life. A pinpointed HPC jet, directed exactly to the cutting zone, effectively penetrates this film, and overcomes the unwelcome obstacle. It also improves the cutting action by changing the shear-plane angle and creating thin manageable chips. High-velocity coolant flow removes the chips.

This significantly improves chip evacuation and prevents chip re-cutting It allows the design of cutters with smaller chip gullets, leading to a higher number of cutter teeth. Additionally, reducing the temperature in the cutting zone enables an increase in the width of the cut. Overall, HPC provides a good solution for higher cutting speed and feed rate, thereby boosting productivity. However, taking advantage of high pressure cooling techniques is only possible when using appropriate machine tools or by modernizing existing machines.

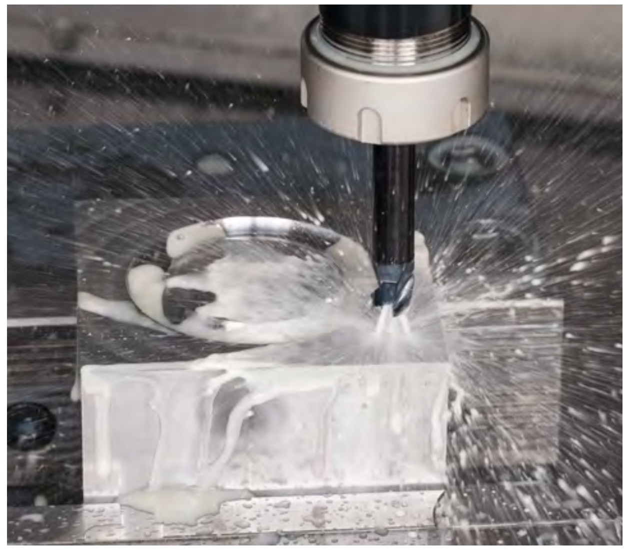


Fig. 50. A pinpointed high-pressure coolant jet significantly improves performance of a MULTI-MASTER endmill when machining hard-to-cut aerospace materials

Dry Machining and Other Options

Ignoring cases where the use of cutting fluid is essential, machine operators must appreciate that if the use of wet cooling brings disadvantages, eliminating coolant will result in noticeable progress. However, dry machining offers promising opportunities. As previously explained, indexable rough milling with significant stock removal results in extremely high heat generation. In this situation, a coolant supply may be destructive due to critical thermal stress. In contrast, when dry rough indexable milling, if the machining data is set correctly, the temperature of the insert's cutting edge will remain high but not excessively high, staying within an acceptable level. For example, the tool temperature will vary within a relatively narrow range, such as 300° to 700° C (572° to 1292° F), which will not lead to thermal shock. For light cuts of high speed milling (HSM), especially for workpieces with hardness values of HRC 45 and above, cooling by air is strongly recommended. In the above examples, the absence of wet coolant also considerably increased tool life.

Fig. 51. Wet cooling is not recommended for heavy-duty milling of steel with indexable tools.

Other important factors to consider are cooling economy and work safety. If cutting tool investment in batch production is estimated at 3% of a part's cost, the share connected with wet coolant (purchasing, maintaining, filtration, etc.), according to a variety of sources, can reach 16-17%. Additionally, prolonged exposure to wet coolant by operating personnel may cause health problems and industrial illnesses.

Many national and international standards and published advice relating to safety and environmental control make increasingly tougher demands related to cutting fluids. Where there is no cutting fluid, there is no need for a coolant pump, a coolant recycling system, and other expensive machine tool accessories, further reducing total costs. The above points ensure that informed manufacturers are constantly looking for alternatives to traditional cutting with coolant supply.

Another available option is milling with minimum quantity lubrication (MQL), sometimes called “near-to-dry”. When using this technique, the tool’s cutting edge works inside a mist formed from oil and compressed air that is sprayed directly into the cutting zone. Depending on the design of a machine tool and milling cutter, the mist can be delivered externally or internally (via the cutter). The main function of MQL is to lubricate the edge during the cutting action, because of this, the machining process consumes only the necessary quantity of oil, and therefore the lubrication is more effective. In addition, the resulting machined workpiece and chips are almost dry (“near-to-dry”), making their cleaning much easier and quicker. MQL increases tool life. Moreover, the working area of the machine tool also remains relatively dry, enabling various parts of the machine tool to work under better conditions and improving their effective life.

One more coolant option is cryogenic machining. Using a coolant at extremely low, cryogenic temperatures drastically reduces the possibility of overheating and allows better performance and extended tool life. Combining this principle with MQL results in a more effective “minimum quantity” cryogenic machining method, as low-temperature coolant (such as liquid nitrogen) is supplied directly to the cutting zone via the tool. Alternatively, some processes propose applying carbon dioxide (CO_2) that is delivered under pressure to the cutting zone. In each of these methods, the particles of cryogenic coolant vaporize from the tool edge, and in doing so, remove heat. However, it is obvious that despite the clear benefits, cryogenic cooling is not a cheap method, and it also requires the use of specially designed machine tools.

The Milling Tool

So - dry or wet? As we can see, the correct answer today continues to be dry and wet – it depends on the specific application (a workpiece material, operation, etc.) and available machining tools. Nevertheless, the manufacturers of cutting tools consider customer requirements and provide them with tools that will ensure productive machining with the use of different cooling methods.

The vast majority of modern indexable mills have internal channels enabling the supply of coolant directly via the tool body. This allows more effective delivery of the coolant directly to the cutting zone. For face mills of previous generations, without coolant channels, **ISCAR** proposes a clamping screw with an adjustable nozzle – in many cases it not only improves coolant supply but also contributes to better chip evacuation. When exploring milling cutters intended for high pressure coolant (HPC) and cryogenic machining, the body of the cutter should be designed accordingly. The shape of the internal channels, their size and sealing elements (if necessary) should ensure the maximal free flow of coolant without any disturbance. Additive manufacturing (AM) technologies enable maximum use of computational fluid dynamics for optimizing the profile of inner channels to ensure efficient pinpointed coolant through the cutter body. Traditional machining processes have limitations in shaping the channels, while AM capabilities provide endless options.

Additional very important elements are the nozzles that are mounted in the outlets of the channels, as they optimize the effect of the high-velocity coolant jet and direct it exactly to the necessary area.

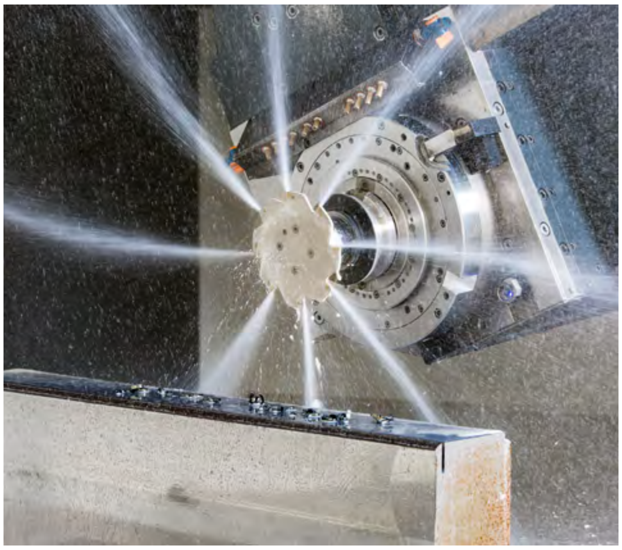


Fig. 52. ISCAR's TANGSLIT indexable slitting cutters with inner coolant channels directed to each and every cutting edge is suitable for use with both standard coolant and high-pressure coolant applications.

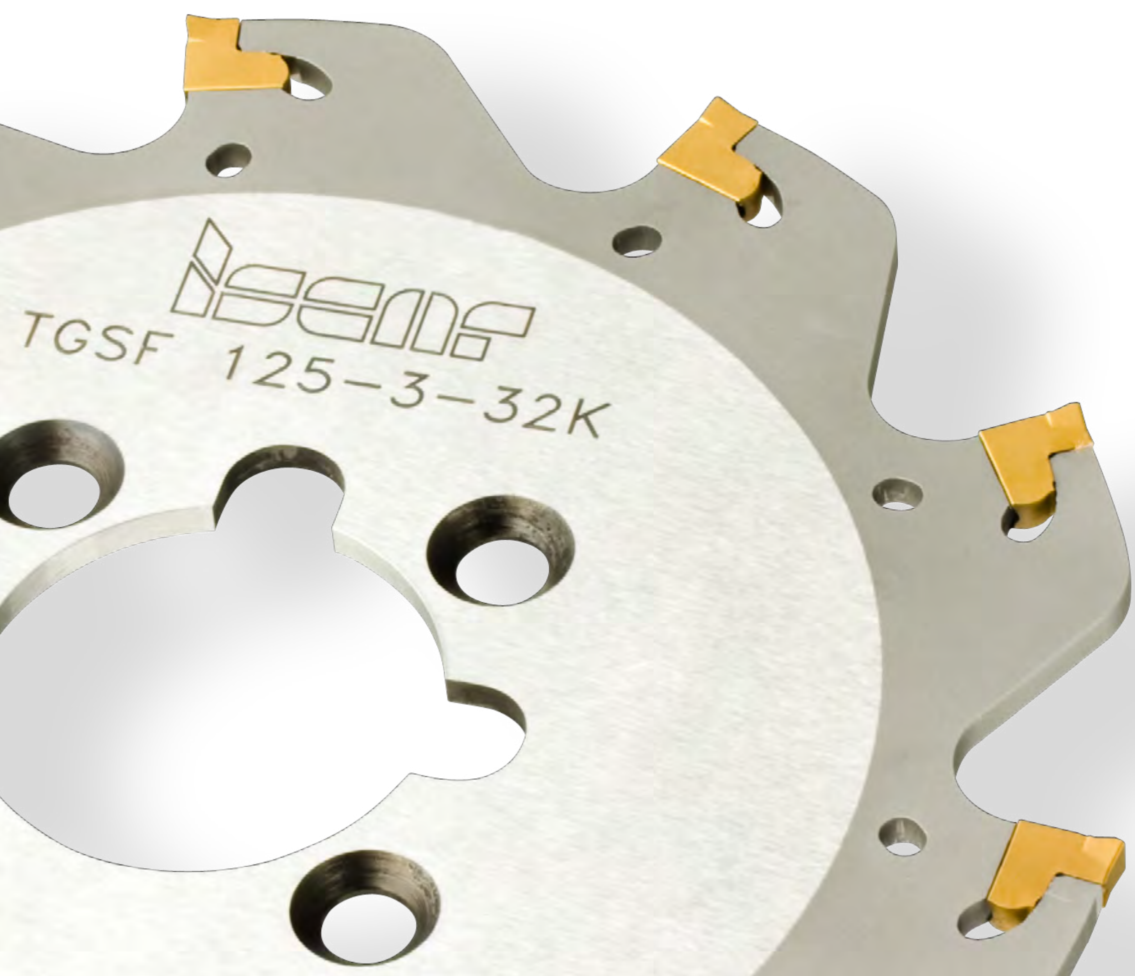
Why Are Nozzles Used As Coolant Outlets

in indexable milling cutters with high pressure coolant (HPC) option?

There are two reasons for using nozzles as coolant outlets: technological and applicative. HPC supply through the body of a cutter requires small-diameter outlets (as well as demands regarding the shape). As manufacture of the outlets via drilling hard steel tools would encounter technological difficulties, screw-in nozzles represent a more practical option.

In indexable extended flute milling tools, if a depth of cut is smaller than the maximum cutting length, there is no need to supply coolant to the inserts that are not involved in cutting. To improve performance, you can easily unscrew the appropriate nozzles from their holes, and then close the hole by a plug or a standard set screw.

Last but not least – we must consider the indexable carbide insert itself. Although the insert's edge performs the cutting, how does it relate to the coolant method? The key to understanding this relationship is the insert's carbide grade and more specifically - its coating, which provides a barrier for heat penetration. The coating must be resistant to the thermal shock that causes the destructive effect. Understandably, there is no "universal" coating, which is equally suitable for productive milling with coolant and without it. Some coatings are more effective for wet machining, whilst others provide dry machining advantages. Although indexable carbide inserts are available with coatings to suit all applications, the field of insert coating layers is so complex it is worth an entirely separate discussion.



Face Milling For High Surface Finish

In the metalworking industry, ensuring a high-grade surface finish has always been one of the main directions in cutting tool development. This is also true for face milling cutters, especially for the indexable ones.

When considering an indexable milling cutter, the key factors that determine the quality of a generated surface are the cutter's geometry and accuracy. Both characteristics are mainly related to inserts carried by the cutter.

Powder metallurgy processing has enabled producing carbide inserts with complex shapes, ensuring optimal cutting geometry while maintaining substantially increased accuracy of inserts as sintered inserts. Not surprisingly, such advancements in technology have significantly improved the surface finish in face milling. However, when indexable milling cutters were successfully applied to machining high-strength materials, and manufacturers started to notice hard milling as an alternative to grinding, the metalworking industry started demanding higher surface finish grades provided by milling.

An indexable face milling cutter is a multi-tooth tool. More teeth = more productivity. This is an undeniable advantage of the cutter. But, in terms of surface texture, a large number of teeth may be a problem. A fine distinction in teeth protrusion leads to irregular feed for the teeth and contributes to chatter, which will ultimately affect the surface finish.

It is perfectly clear that insert accuracy can considerably be increased by grinding. Moreover, grinding provides a sharp cutting edge that is very important in maintaining cutting action and preventing plastic deformation of metal in fine milling, which features shallow depths of cut. Ensuring a highly accurate cutting edge requires grinding both the top and side surfaces of an insert. This may cancel the advantages of powder metallurgy in generating complicated surfaces to provide required rake and clearance angles along a cutting edge. To avoid such an adverse impact, tool engineers should be very resourceful when designing the inserts intended for fine grinding.

An important factor regarding tooth accuracy loss is the insert pocket in a cutter has its own dimensional and form tolerances. As a result, even for ideally precise inserts, teeth protrusion will vary within acceptable limits, although it cannot ensure extra fine surface finish when compared to grinding. A way to overcome this problem is by using a fly cutter that carries only one insert. The fly cutter that is successfully used in various milling applications facilitates a smooth and clean cut, providing excellent surface texture parameters. But then again, productivity in this case is far below multi-tooth indexable face mills.

How to solve a difficult situation and find an acceptable balance between surface quality and productivity? The Cutting tool manufacturers have developed different ways to answer.

An integrated wiper flat with a specially shaped minor cutting edge is a classical element of various milling inserts. Its width should be greater than the feed per revolution. Despite being called “flat”, the minor edge sometimes has a complex geometry to compensate for a negative effect of wear development. When an insert is mounted on a cutter, the wiper flat sits parallel to the machined surface. Hence, the surface will be formed by the most protruding insert of the cutter. Introducing a wiper flat in an insert design is an effective way to improve surface finish. And even today, rough milling inserts may have an integrated wiper (Fig. 53). These inserts being mounted on milling tools that are intended for rough and semi-rough operations, can significantly improve surface finish providing surface roughness, which usually feature semi-finish to finish milling passes. For example, when milling steel and cast iron, the roughness parameter Ra is estimated up to 0.4 µm (16 µin).

Fig. 53. ISCAR’s NEODO double-sided square rough milling inserts utilize an integrated wiper flat to provide better surface finish.

Increasing the number of teeth in big-diameter face mills and fine pitch cutters determines the appropriate growth of the integrated wiper width, which has a natural bound due to design and dimensional limitations. In such cases, a high surface finish can be achieved with the use of a specially designed wiper insert (or two inserts for large-sized tools), whereby the wiper flat is significantly wider than the standard one. This insert is mounted in the same pocket but protrudes several hundredths of a millimeter (thousandths of an inch) axially relative to the standard insert, usually 0.05-0.07 mm (.002-.00275”) (Fig. 54).

Semi-Roughing and Semi-Finishing in Milling

The difference between these two definitions can be blurred, and they may often be considered synonyms. However, in some cases when milling a surface requires more than one operation, these operations are specified as rough milling, semi-rough milling, semi finish milling, finish milling, fine milling or simply roughing, semi roughing, semi finishing, finishing.

The same situation may be observed not only in milling but also in other types of machining, such as turning.

Very good results can be reached by applying adjustable milling cutters that utilize different mechanisms to adjust the position of an insert cutting edge within very strict limits (only several microns). But the beneficial adjustability of cutting tools also has a flip side as well; it is spider work, which takes time.

A desirable solution looks like a tool that after mounting an insert has no adjusting requirements needed to achieve high surface quality grade. That is why improving accuracy and advanced geometries remain the mainstream in updating indexable cutters for finish face milling.

Wiper Flat and Wiper Insert

A wiper flat is a small minor edge on a regular indexable insert in milling cutters to improve the quality of a machined surface. It is also referred to as a “wiper” or, sometimes, a “parallel land.” The term “parallel” highlights that this land, the minor cutting edge of an insert, is generally parallel to the machined surface.

A wiper insert is a specially designed insert where the wiper flat is significantly larger than for a standard insert.

When mounted in a milling cutter, the wiper insert protrudes 0.05...0.07 mm (.002-.00275”) axially relative to a regular insert. A wiper insert “smooths down” the machined surface, noticeably improving surface finish.

Fig. 54. A wiper insert mounted on a face mill.

Despite significant improvement in the high surface quality of milled surfaces, tool designers still believe that available resources are not yet exhausted and that intelligent applications of the latest generations of advanced milling cutters can substantially improve surface texture. Cutting tool manufacturers offer unique solutions that attract attention with their originality. An example is **ISCAR**'s **TANGFIN** family of step milling cutters carrying tangentially clamped indexable inserts with wide integrated wiper flats (Fig. 55).

The inserts are positioned in a **TANGFIN** cutter with a gradual displacement in both radial and axial directions, and therefore, each insert cuts a small section of the machined material providing an extra fine surface finish with roughness Ra up to 0.1 µm (4 µin).

Fig. 55. ISCAR's TANGFIN indexable milling cutter utilizes the principle of graduated insert displacement.

A tendency to decrease machining allowance due to the active introduction of technologies for precise workpiece production and 3D printing makes the issue of obtaining a high surface finish by face milling particularly relevant. Can toolmakers find a prompt, simple, and effective answer to the new needs of manufacturing? The near future will tell.

Step Milling Cutter

A step milling cutter is a type of mill with teeth that are equally spaced relative to each other in either the axial or radial direction. If the teeth are formed by indexable inserts, the cutter is referred to as an indexable step milling cutter.

Self-evaluation quiz

27- Does ISCAR produce tools for milling non-ferrous metals like aluminum and copper?

a. Yes.

b. Now.

28- Do ISCAR's milling tools with exchangeable heads include the heads with indexable inserts?

a. Yes..

b. No.

29- Wet coolant facilitates

a. Cooling.

b. Both cooling and lubrication.

c. Cooling but the main function of wet cooling is ensuring normal chip flow.

30- In milling with minimum quantity lubrication (MQL), the coolant is

a. Compressed air.

b. Low-temperature carbon dioxide (CO_2).

c. High-pressure fluid.

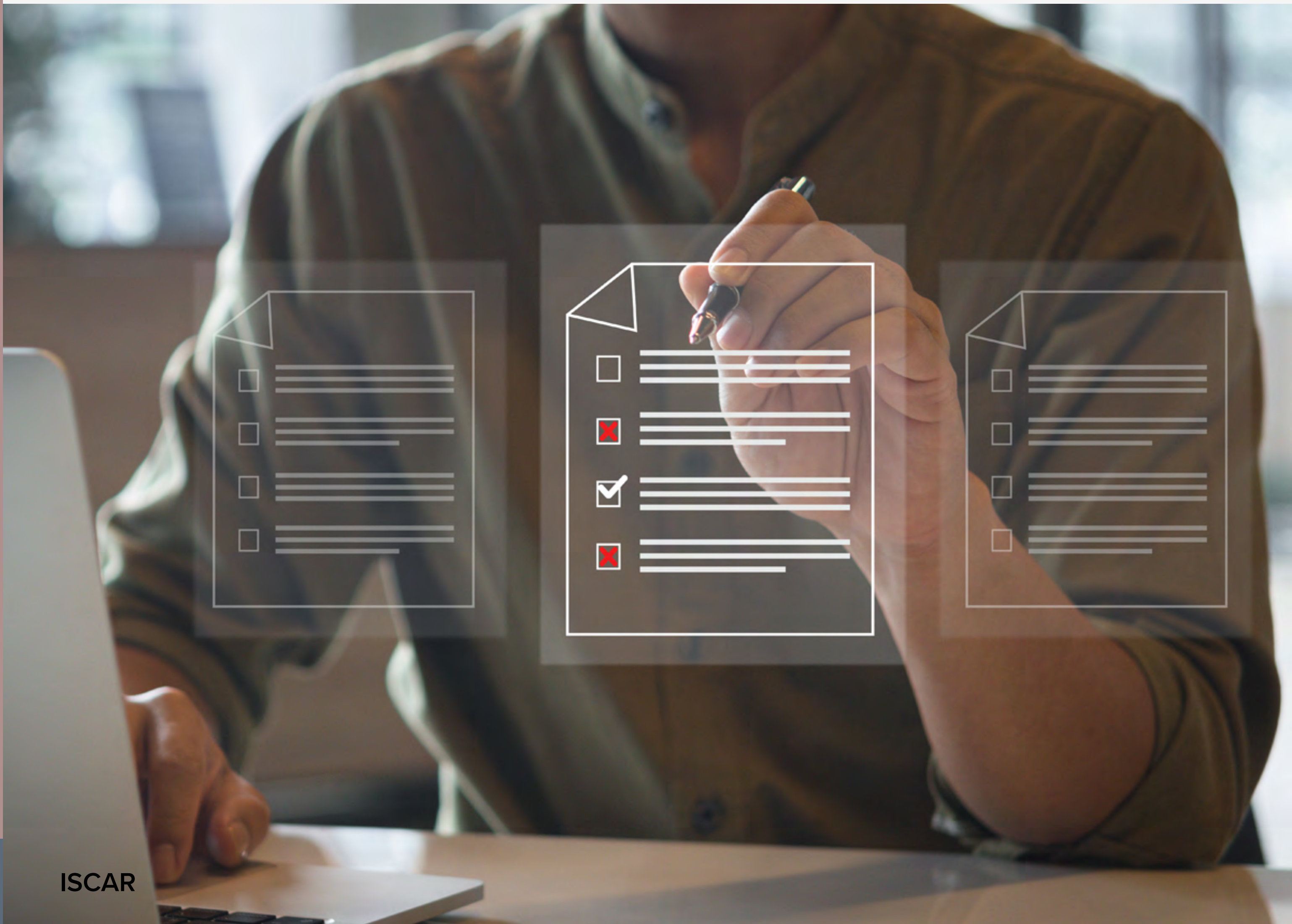
d. A mist from compressed air and oil.

31- A wiper flat is

a. The major cutting edge of an insert to improve surface finish.

b. Another name for a wiper insert.

c. The minor cutting edge of an insert to improve surface finish.



How to Select The Right Milling Tool

From a broad perspective, the primary factor to consider when selecting a milling tool is the cost of the machining operation performed by the tool and its impact on the cost per unit (CPU) for a machined part as a whole. Even though the cost of the tooling has a minor share in the CPU, its indirect influence on reducing the CPU can be significant. This is because the tool, a small part of the technological equipment necessary for the manufacturing process, can sometimes be the only obstacle preventing a milling machine from running fast and thus reducing machining time. Therefore, the most efficient milling tool should be used to increase productivity and lower the CPU.

Another important aspect to consider is the versatility of the tool or its ability to effectively perform various milling operations such as shoulder milling in combination with ramping and plunging. Using one tool for different applications, when machining a part, not only shortens time needed for tool changes but also reduces the amount of tool stock in a shop warehouse. The versatility can be further increased by using milling tools with exchangeable cutting heads, which render for a head change when the tool or its holder is clamped into a machine spindle without requiring additional setup time.

In conclusion, when selecting a milling tool, it is crucial to consider these obvious but often overlooked points. More specifically, the Application-Geometry-Grade (AGG) chain analysis should be applied. This involves answering commonly known checkpoints that guide the selection of the most suitable milling tool:

Application

What is the type of a machining operation?

Workpiece: its material, hardness and roughness before the operation.

Required accuracy and surface finish.

Machining allowance (stock to be removed during operation).

Machining strategy.

Type of machining (light, moderate, heavy).

Tool adaptation (shank-type, shell mill).

Operation stability (high, normal, poor).

Machine tool limitations (power, torque, spindle speed).

Coolant (dry, wet, HPC etc., possibility of coolant through spindle).

Geometry

Which cutting geometry is recommended for machining the workpiece for the above requirements? (All types of milling tools – indexable, solid, with exchangeable heads – should be considered).

Grade

Which grade of a cutting tool material is more suitable for machining the workpiece for the above requirements.

When deciding on the appropriate cutting tool for a particular application, it is difficult to determine whether a standard tool or a special (tailor-made) tool is preferable. The ideal tool selection is contingent on various factors, such as the nature of the business situation, the manufacturing program, the production type, and sometimes personal preferences. Standard cutting tools, produced by a specific tool manufacturer, offer high versatility and are appropriate for machining a diverse range of parts that come in different shapes.

Furthermore, the tool exhibits excellent performance capabilities when cutting various engineering materials. To ensure seamless production processes, it is crucial to have the cutting tools delivered in a timely manner. This is why standard tools are the foundation of tool stock management on metalworking production floors.

A special cutting tool is designed for specific operations on a particular part, made of a specific material, and used on a machine that requires a specific work holding fixture. This custom-engineered tooling solution aims to provide the best possible performance and outcome. However, there is a downside to this solution as it limits the tool's versatility, making it less adaptable to different applications. As a result, special tools are primarily used for high-volume mass production, especially in the automotive industry.

A special tool is not readily available and requires significant engineering effort, including concept design, coordination with the customer, detailed design, and production.

The delivery time for special tools is significantly longer than that of standard tools. Metalworking shops often face a dilemma when choosing between standard or special tools, which can impact project timelines. The question remains: which tool will provide the best solution? Should shops rely on readily available standard tools or opt for a highly efficient special tool with a longer delivery time? Naturally, economic factors such as cost per unit and tool costs should be taken into consideration. All things being equal, in an ideal situation where production programs, processes, and inventory are planned, the answer is clear. Considering lead time is an important factor when choosing special tooling.

Why Are a Significant Number of Milling Cutters with High-Pressure Cooling (HPC) Specially Tailored?

The main consumers of HPC milling cutters are manufacturers working with hard-to-cut materials, for example titanium alloys. In many cases, producing parts from the materials requires a high volume of metal removal. To boost productivity, manufacturers often use unique machine tools, and, to reach maximum operational rigidity, they prefer integral tools with direct adaptation to the spindle of a machine - without intermediate tooling such as arbors or holders. Specific tool diameters, cutting lengths, and overhang, as well as adaptations that vary from one manufacturer to another, demand tailor-made HPC milling cutters.

Special tools vary in their design complexity. Some are simple modifications of standard tools, such as changes to the corner radius or tool length. These modifications fall under the category of “semi-standard” products, which can be manufactured relatively quickly. The design complexity of a special tool is determined by a pre-design study that assesses the customer’s manufacturing limitations, accompanied by cost calculations and production time. The results of the study determine the limits and cost-effectiveness that correlate to the special tool’s delivery time. However, there are additional ways to reduce the delivery time of special tools, such as using modular assembled tools or solid tools with exchangeable cutting heads. The tool manufacturer’s delivery times and production abilities play a significant role in the final decision on how to proceed.

NEOITA – Your Assistant in the Selection of the Required Milling Tool

If the application parameters are known, NEOITA (**ISCAR**’s Tool Advisor), a computer-aided search engine, can be a very effective tool. This expert-system software, which is is based on engineering knowledge and practical experience, enables the user to find the most suitable tool for a specific machining operation, including milling. The software is available in various languages and can be accessed via smartphone Apps.

New Benchmarks in Milling

These days, it is hard to imagine any machine shop without milling machines on the shop floor. Milling, as an essential process in manufacturing, is an integral part of machining technology, driven by the increasing demands of manufacturing. However, there are specific aspects that uniquely impact the advancement of milling.

Today, we are witnessing significant changes in manufacturing that will have a profound impact on milling technologies of the future (machine, cutting tools, CAM programming, etc.). These changes are driven by various factors, such as the increasing accuracy of metal shaping through precision investment castings and precision forging, the widespread adoption of 3D printing, the growing usage of new composite and sintered materials, the need to enhance productivity in machining hard-to-cut superalloys and titanium grades, and the strong focus on electric and hybrid cars in the automotive industry, contributing to these changes. In addition, advancements in multi-axis machine tools have opened up new possibilities for precise machining of complex parts and have enabled the implementation of new cutting strategies to improve productivity. In modern technological processes, there is a tendency to significantly reduce the amount of machining stock intended for milling operations, while simultaneously increasing the requirements for surface finish and accuracy.

Therefore, the advancement in milling is driven by the need for higher productivity, more precision, and sustainability in milling operations. Consequently, the main developments in milling can be characterized as follows:

1- Fast metal removal focuses on boosting the metal removal rate (MRR) to achieve higher productivity by significantly increasing cutting speed or feed per tooth. This is achieved through techniques such as high speed milling (HSM) and, in rough operations, high feed milling (HFM).

2- Precision milling provides higher accuracy in milling operations.

3- Multi-axis milling is characterized by the utilization of multi-axis machining centers to enable complex milling operations.

4- Adaptive milling aims to develop intelligent milling systems that can adapt to changing conditions during the machining process.

5- Sustainable milling strives to reduce the environmental impact of milling operations. It involves the development of eco-friendly cutting fluids, recycling and reusing of materials, and the use of energy-efficient machine tools and milling cutters.

The success in these areas relies on the synergy of several key components, namely machine tools, cutting tools, and computer-aided engineering (CAE) systems. High-speed milling, for instance, necessitates machine tool technologies capable of handling exceptionally high rotational velocities, as well as advanced cutting materials and coatings for milling tools.

Simultaneously, enhancing the precision of milling operations requires not only milling cutters with tighter tolerances but also improved control systems and linear motor drives. In the case of multi-axis milling, the breakthrough lies in the addition of more effectively controlled axes of movement,

along with the application of appropriate cutting geometries for milling tools. Adaptive milling, on the other hand, incorporates innovations such as the use of state-of-the-art monitoring systems, high-sensitive sensors, and efficient algorithms to optimize cutting data and tool paths in real-time. Moreover, sustainability advancements require energy-efficient milling strategies that employ suitable machine tools, cutting tools, and eco-friendly coolant techniques.

Indexable milling reflects the ways of advancement that feature exchangeable cutting inserts in machining operations:

a. Advanced insert materials is an ongoing process to improve the cutting materials for indexable milling inserts including the development of advanced carbide grades, ceramics, and ultra hard cutting materials.

b. Coating technologies with continuous R&D focuses on new coatings to improve wear and heat resistance while enhancing lubricity.

c. Progressive cutting geometry optimizes cutting geometry and chip forming topology of inserts to improve cutting action, diminish cutting forces, and enhance chip flow in milling operations.

d. The effective utilization of cutting material incorporates intelligent insert design to provide maximum indexable cutting edges without reducing cutting capabilities.

e. In addition, the distinct course on smart manufacturing requires the integration of digitization into milling operations and milling tools. Referring to milling tools, digital twins and appropriate software applications have already become the "must" features of a comprehensive tool range.

Answers for self-evaluation quiz*
(*The right answers are highlighted by green)

1- What is the primary motion in milling?
   a. **Rotary motion of a mill.**
   b. Rotary motion of a workpiece.
   c. The primary motion can be utilized with both above motions.
   d. Linear motion of a workpiece.

2- The feed motion in milling is
   a. The linear motion of a machined workpiece.
   b. The rotary motion of a machined workpiece.
   c. **The translational motion of a machined workpiece relative to the cutter.**
   d. The rotary motion of a mill.

3- The cylindrical mill is intended for
   a. **Peripheral milling.**
   b. Face milling.
   c. End milling
   d. Both peripheral and face milling.

4- Side-and-face milling relates to
   a. Milling plane face bounded by shoulder.
   b. End milling.
   c. **Milling slots and groove by disc milling cutter.**
   d. Milling complex 3D surfaces

5- Should up milling, which ensures a better cutting effect, be considered as the first-choice type of milling, and applied wherever possible?
   a. Yes, it is correct.
   b. **No, it is not correct.**

6- Is the combination of up and down milling typical for most face milling operations?
   a. **Yes, it is correct.**
   b. No, it is not correct.

7- What are the main elements of a milling cutter?
   a. **A body and a cutting part.**
   b. A body, a cutting part, and inner channels for coolant supply.
   c. A body, a specially shaped area of the body that forms a cutting part, and a specially created chip space between the cutter teeth to ensure the chip flow.

8- Cutting geometry of a milling cutter is mainly determined by
   a. The cutting edge inclination and the cutting edge angle.
   b. **The set of appropriate angles.**
   c. The directions of the radial and axial rakes.

9- The cutting edge angle is also known as
a. The approach angle.
b. The lead angle.
c. The dish angle
d. **The entering angle.**

10- The lead angle is the angle complementary to
a. The cutting edge inclination.
b. The approach angle.
c. **The cutting edge angle.**
d. The minor cutting edge angle.

11- The flutes of a right-hand cutting solid endmill can be
a. Helical with a right-hand helix.
b. Helical with a left-hand helix.
c. Helical with right- or left-hand helix.
d. **Helical or straight but it is more common for designs to feature helical flutes.**

12- Do feed per tooth and feed rate mean the same?
a. Yes, these terms are the equivalent.
b. **No, these terms are different.**

13- The effective diameter of a profile milling cutter is
a. The nominal diameter of the cutter.
b. The largest diameter of the cutter.
c. **The largest true cutting diameter of the cutter.**

14- Is the maximum chip thickness always equal to the feed per tooth?
a. Yes, this is correct.
b. **No, this is not correct.**

15- In milling, the necessary power consumption and cutting torque are calculated based on
a. The tangential cutting force.
b. The total (resultant) cutting force.
c. The bending force.

16- Chatter is
a. **A specific type of vibrations that characterize machining processes.**
b. The type of specific forced vibrations, caused by the variation of a tangential cutting force.
c. The type of vibrations that occur when the tooth of a milling cutter periodically enters machined material and leaves it.

17- Today in shop-floor conditions, the most common practice in trying to reduce vibrations in milling is
a. Using anti-vibration tool holders.
b. Computer modelling of the process with the appropriate real-time response.
c. **Changing cutting data.**

18- Can the cutting geometry of a milling cutter be a design tool of anti-vibration solution?

a. **Yes.**

b. No.

19- Does the material of a cutter body enable diminishing vibrations in some milling applications?

a. **Yes.**

b. No.

20- High speed machining (HSM) is characterized by

a. Shallow cuts combined with high feed per tooth.

b. Shallow cuts and significant width of cut combined with high rotational velocity of a mill.

c. **High-speed rotation of a mill combined with shallow cuts.**

d. High rotational velocity of a mill combined with a feed motion along the mill axis.

21- Trochoidal milling is a method of

a. **High speed machining.**

b. High feed milling.

c. Plunge milling.

d. Turn-milling.

22- Dynamic milling is suitable for every high-speed machine tool.

a. Correct.

b. **Not correct.**

23- High feed milling concept utilizes chip thinning effect.

a. **Correct.**

b. Not correct.

24- High feed milling cutter features

a. **Small cutting edge angles.**

b. Small lead angles.

25- Plunge milling is suitable for

a. **Roughing.**

b. Finishing.

c. both roughing and finishing.

26- Face turn-milling is not suitable for machining non-continuous surfaces.

a. Correct.

b. **Not correct.**

27- Does **ISCAR** produce tools for milling non-ferrous metals like aluminum and copper?

a. **Yes.**

b. Now.

28- Do **ISCAR**'s milling tools with exchangeable heads include the heads with indexable inserts?

a. **Yes.**

b. No.

29- Wet coolant facilitates

a. Cooling.

b. **Both cooling and lubrication.**

c. Cooling but the main function of wet cooling is ensuring normal chip flow.

30- In milling with minimum quantity lubrication (MQL), the coolant is

a. Compressed air.

b. Low-temperature carbon dioxide (CO_2).

c. High-pressure fluid.

d. **A mist from compressed air and oil.**

31- A wiper flat is

a. The major cutting edge of an insert to improve surface finish.

b. Another name for a wiper insert.

c. **The minor cutting edge of an insert to improve surface finish.**

ISCAR LTD.
Israel
Headquarters
Tel +972 (0)4 997 0311
Fax+972 (0)4 987 3741
headquarter@iscar.co.il

Argentina
ISCAR TOOLS ARGENTINA SA
Tel +54 114 912 2200
Fax+54 114 912 4411
admin@iscararg.com.ar

Australia
ISCAR AUSTRALIA PTY. LTD
Tel +61 (0) 2 8848 3500
Fax+61 (0) 2 8848 3511
iscaraus@iscar.com.au

Austria
ISCAR AUSTRIA GmbH
Tel +43 7252 71200-0
Fax+43 7252 71200-999
office@iscar.at

Belarus
JV ALC "TWING-M"
Tel/Fax: +375 17 511 82 91
Tel/Fax: +375 17 511 82 93
Tel/Fax: +375 17 511 82 95
info@twing.by
www.twing.by

Belgium
n.v. ISCAR Benelux s.a.
Tel +32 (0) 2 464 2020
Fax+32 (0) 2 522 5121
info@iscar.be

Bosnia
(Representative Office)
Tel +387 32 201 100
Fax+387 32 201 101
info@iscar.ba

Brazil
Iscar do Brasil Coml. Ltda.
Tel +55 19 3826-7100
Fax+55 19 3826-7171
DDG 0800 701 8877
iscar@iscarbrasil.com.br

Bulgaria
ISCAR BULGARIA
Tel/Fax +359 431 62557
aa_iscar@infotel.bg

Canada
ISCAR TOOLS INC.
Tel +1 905 829 9000
Fax+1 905 829 9100
admin@iscar.ca

Chile
J&A INTERNATIONAL
Tel +56 2 2232 5838
amedina@jya.cl
www.jya.cl

China
ISCAR CHINA
Tel +86 21 8024 8888
iscar@iscar.com.cn

Colombia
ISCAR Andina
Tel +57 310 380 9932
Tel/fax +57 1 896 65 78
iscar@iscar.com.co

Croatia
ISCAR ALATI d.o.o.
Tel +385 (0) 1 33 23 301
Fax+385 (0) 1 33 76 145
podrska@iscar.hr

Cyprus
WAMET (Demetriades) Ltd.
Tel +357 (0) 2 336660/5498
Fax+357 (0) 2 333386
wamet@cytanet.com.cy

Czech Republic
ISCAR CR s.r.o.
Tel +420 377 420 625
Fax+420 377 420 630
iscar@iscar.cz

Denmark
KJ VAERKTOEJ AS/ISCAR DENMARK
Tel +45 70 11 22 44
Fax+45 46 98 67 10
kj@kj.dk

Ecuador
ISCAR Andina
Tel/fax +57 1 821 93 38
iscar@iscar.com.co
atencioncliente@iscar.com.co

Estonia
KATOMSK AS
Tel +372 6775 671
Fax+372 6720 266
aleksei@katomsk.ee

Finland
ISCAR FINLAND OY
Tel +358-(0)9-439 1420
Fax+358-(0)9-466 328
info@iscar.fi

France
ISCAR FRANCE SAS
Tel +33 (0)1 30 12 92 92
Fax+33 (0)1 30 12 95 82
info@iscar.fr

Germany
ISCAR Germany GmbH
Tel +49 (0) 72 43 9908-0
gmbh@iscar.de
www.iscar.de

Greece
INTERNATIONAL TOOLS
K.-X. GEORGOPOULOS & SIA O.E
Tel +30 210 346 0133
Fax+30 210 342 5621
info@internationaltools.gr

VIMA
V. Mazloumian & Sons
Tel +30 2310 517-117 / 544-521
Fax+30 2310 529-107
vimaco@otenet.gr
http://www.vimaco.gr

Hong Kong
MTC TOOLING SYSTEMS LTD
Tel +85-2-23054838
Fax+85-2-27988789
yoongkamsing@hotmail.com

Hungary
ISCAR HUNGARY KFT.
Tel +36 28 887 700
Fax+36 28 887 710
iscar@iscar.hu

India
ISCAR India Ltd.
Tel +91 77009 63707
sales@iscar.in

Indonesia
CV MULTI TEKNIK
Tel +62-21-29206242/44/45/59
Fax+62-21-29206243
contact@multi-teknik.co.id

Ireland
HARD METAL MACHINE TOOLS
Tel +353 (0) 1 286 2466
Fax+353 (0) 1 286 1514
phannigan@hardmetal.ie
www.hardmetal.ie

Italy
ISCAR ITALIA srl
Tel +39 02 93 528 1
Fax+39 02 93 528 213
marketing@iscaritalia.it

Japan
ISCAR JAPAN LTD.
Tel +81 6 6835 5471
Fax+81 6 6835 5472
iscar@iscar.co.jp

Latvia
MECHA, UAB
Tel +370 37 407 230
Fax+370 37 407 231
info@mecha.lt

Lithuania
MECHA, UAB
Tel +370 37 407 230
Fax+370 37 407 231
sigitas@mecha.lt

Mexico
ISCAR DE MÉXICO
Tel +52 (442) 214 5505
Fax+52 (442) 214 5510
iscarmex@iscar.com.mx

Morocco
2 rue des
Fauvettes Oasis
20103 Casablanca
Tel +212 (0) 7 62 23 48 56
hachem@iscar.ma

The Netherlands
ISCAR NEDERLAND B.V.
Tel +31 (0) 182 535523
Fax+31 (0) 182 572777
info@iscar.nl

New Zealand
ISCAR PACIFIC LTD.
Tel +64 (0) 9 573 1280
Fax+64 (0) 9 573 0781
iscar@iscarpac.co.nz

North Macedonia
(Representative Office)
Tel +389 2 309 02 52
Fax+389 2 309 02 54
info@iscar.com.mk

Norway
SVEA MASKINER AS
Tel +47 32277750
Fax+47 32277751
per.martin.bakken@svea.no

Peru
HARTMETALL SAC
Tel: (511) 6612699
otorres@hartmetallgroup.com

Philippines
MESCO
Tel +63 2631 1775
Fax+63 2635 0276
mesco@mesco.com.ph

Poland
ISCAR POLAND Sp. z o.o.
Tel +48 32 735 7700
Fax+48 32 735 7701
iscar@iscar.pl

Portugal
ISCAR Portugal, SA
Tel +351 256 579950
Fax+351 256 586764
info@iscarportugal.pt

Romania
ISCAR Tools SRL
Tel +40 (0)312 286 614
Fax+40 (0)312 286 615
iscar-romania@iscar.com

Serbia
ISCAR TOOLS d.o.o.
Tel +381 11 314 90 38
Fax+381 11 314 91 47
info@iscartools.rs

Singapore
SINO TOOLING SYSTEM
Tel +65 6566 7668
Fax+65 6567 7336
sinotool@singnet.com.sg

Slovakia
ISCAR SR, s.r.o.
Tel +421 (0) 41 5074301
Fax+421 (0) 41 5074311
info@iscar.sk

Slovenia
ISCAR SLOVENIJA d.o.o.
Tel +386 1 580 92 30
Fax+386 1 562 21 84
info@iscar.si

South Africa
ISCAR SOUTH AFRICA (PTY) LTD.
ShareCall 08600-47227
Tel +27 11 997 2700
Fax+27 11 388 9750
iscar@iscarsa.co.za

South Korea
ISCAR KOREA
Tel +82 53 760 7594
Fax+82 53 760 7500
leeyj@taegutec.co.kr

Spain
ISCAR IBERICA SA
Tel +34 93 594 6484
Fax+34 93 582 4458
iscar@iscarib.es

Sweden
ISCAR SVERIGE AB
Tel +46 (0) 18 66 90 60
Fax+46 (0) 18 122 920
info@iscar.se

Switzerland
ISCAR HARTMETALL AG
Tel +41 (0) 52 728 0850
Fax+41 (0) 52 728 0855
office@iscar.ch

Taiwan
ISCAR Taiwan Ltd.
Tel +886 (0)4-24731573
Fax+886 (0)4-24731530
iscar.taiwan@msa.hinet.net

Thailand
ISCAR Thailand Ltd.
Tel +66 (2) 7136633-8
Fax+66 (2) 7136632
iscar@iscarthailand.com

Turkey
ISCAR Kesici Takim
TIC. VE. IML. LTD
Tel +90 (262) 751 04 84 (Pbx)
Fax+90 (262) 751 04 85
iscar@iscar.com.tr

Ukraine
ISCAR UKRAINE LLC
Tel +38 (050) 440 23 91
info@iscar.com.ua

United Arab Emirates
ISCAR GULF TRADING LLC
Tel +971 4 886 4547
www.svrs-mena.com

United Kingdom
ISCAR TOOLS LTD.
Tel +44 (0) 121 422 8585
Fax+44 (0) 121 423 2789
sales@iscar.co.uk

United States
ISCAR USA
Arlington
USA Headquarters
300 Westway Place
Arlington, TX 76018
Tel +1 817-258-3200
usmarketing@iscarusa.com

Charlotte
Productivity Center
3020 Whitehall Park Drive
Charlotte, NC 28273
Tel +1 817 258-3202
usmarketing@iscarusa.com

Venezuela
FERREINDUSTRIAL ISO-DIN C.A.
Tel +58 2 632 8211/633 4657
Fax+58 2 632 5277
iso-din@cantv.net

Vietnam
ISCAR VIETNAM
(Representative Office)
Tel +84 8 38 123 519/20
Fax+84 8 38 123 521
iscarvn@hcm.fpt.vn

www.iscar.com

Please check ISCAR's online catalog for the most current technical information regarding our products

08/2024
3471284 G

Milling Applications And Cutter Basics

ISCAR's Reference Guide

Member IMC Group

iscar

www.iscar.com