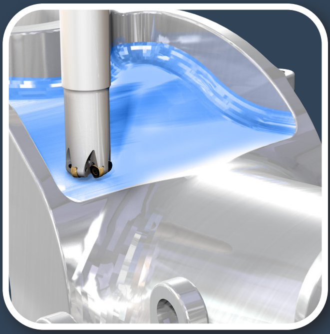
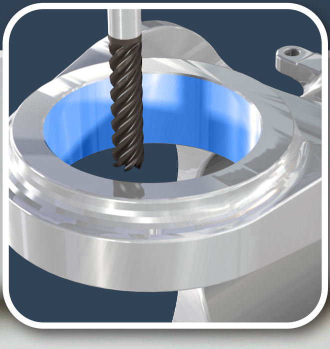
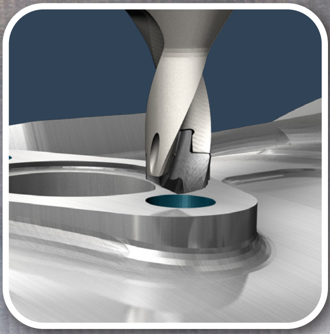
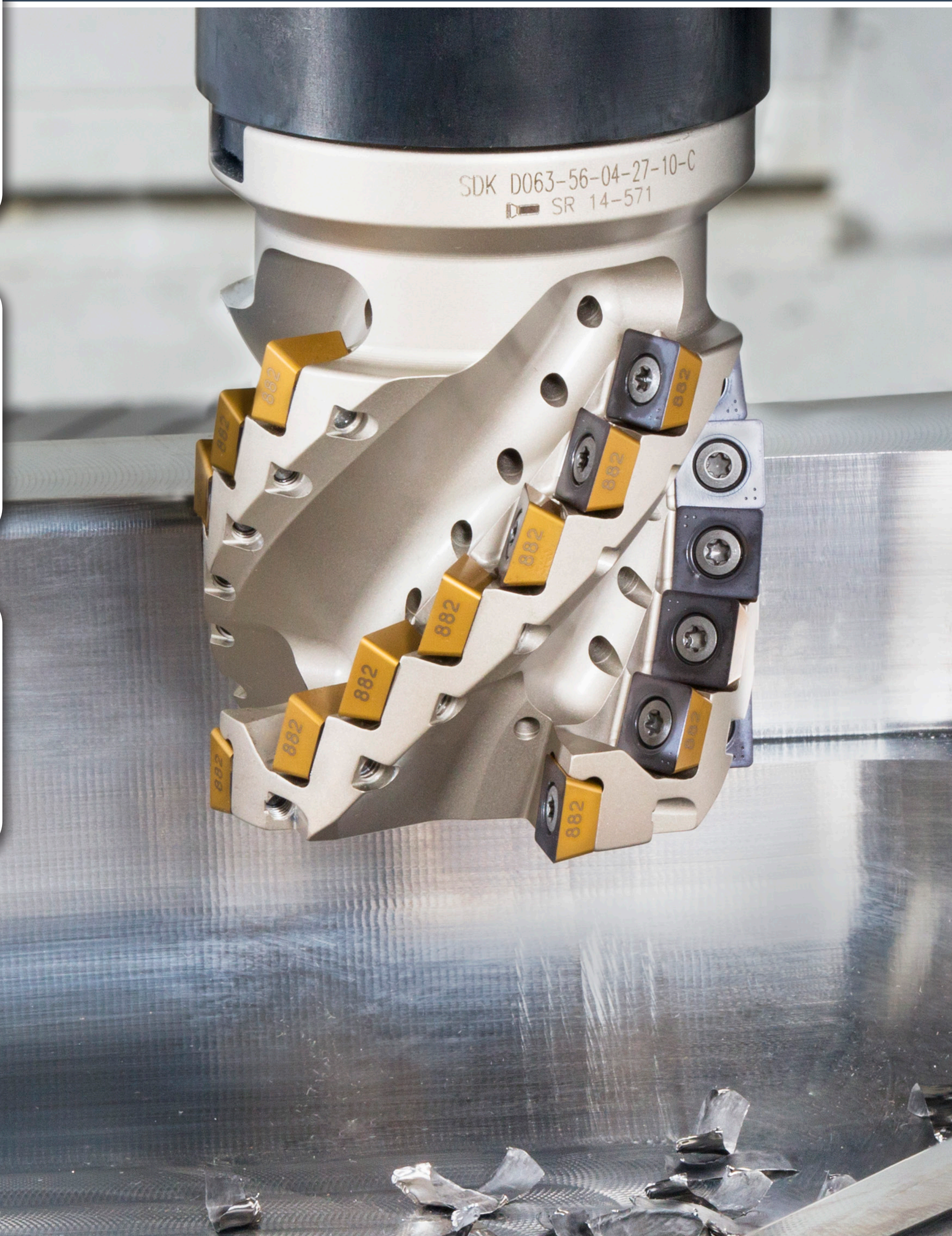
The ISCAR Reference Guide for Machining Titanium









Member IMC Group
www.iscar.com

CERTIFIED QMS AS 9100 THE STANDARDS INSTITUTION OF ISRAEL

CERTIFIED QMS SI ISO 9001:2008 THE STANDARDS INSTITUTION OF ISRAEL

CERTIFIED EMS SI ISO 14001:2004 THE STANDARDS INSTITUTION OF ISRAEL

CERTIFIED OHS SYSTEM SI OHSAS 18001:2007 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.

A WORD TO THE READER

Titanium and its alloys are widely used in modern industry. Today, due to exceptional strength-to-weight ratio, high corrosion resistance and performance over a wide temperature range, these lightweight metals are essential elements in the designs of aerospace, shipbuilding, chemistry and medicine. Aircraft frame components, stressed skins, fasteners, compressor disks and blades, casings, marine screw propellers, and orthopedic implants - these are some typical examples of parts and units made from titanium and its alloys.

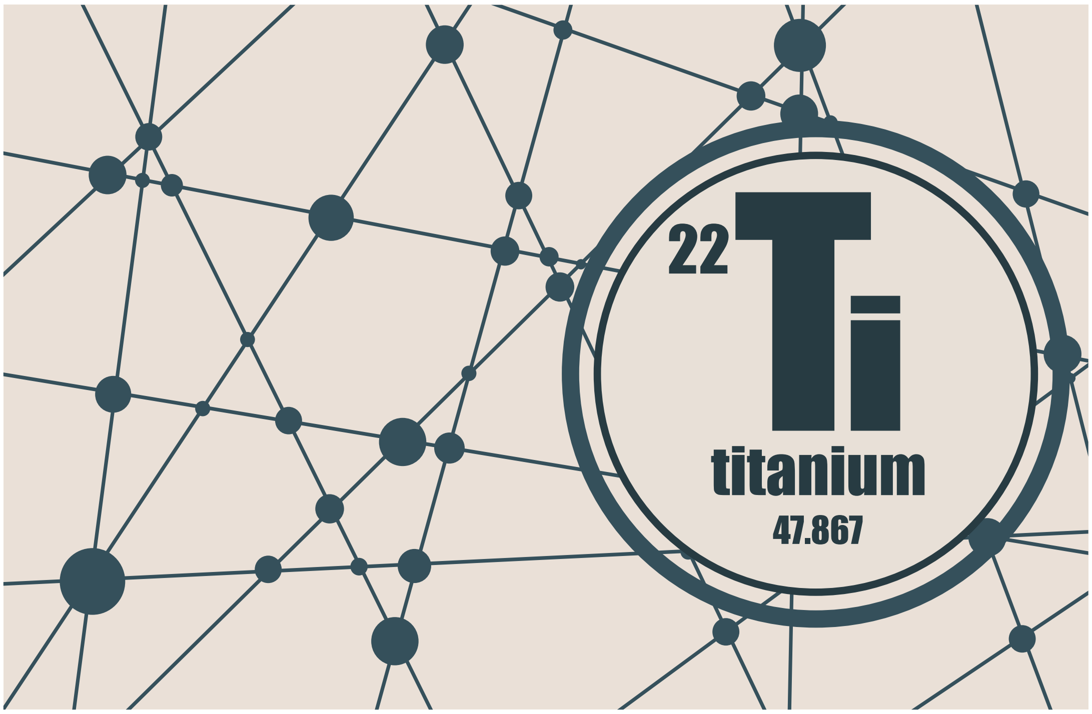
In spite of these indisputable advantages, titanium and titanium alloys are characterized by certain disadvantages; and poor machinability is one of them. Indeed, the Titans were also imperfect– the mythical giants after whom a new lightweight metal, discovered in the late 18th century, was named.

Manufacturers of cutting tools make great efforts to improve their products intended for machining titanium. At ISCAR, we consider the issue of high-performance cutting tools for machining titanium as a primary-importance direction of our work. Our ongoing research and development aim to provide the industry with effective and reliable cutting tools for productive machining of titanium. A guide on how to choose and operate the tools intelligently also contributes to the effectiveness of machining; we hope that this quick reference guide in milling titanium will act as a useful reference tool and a real help in machining titanium with modern ISCAR tools.

The main aim of the guide is to assist in selecting the most suitable rotating tool for machining titanium, and to quickly find initial cutting data required for operating the tool. The guide contains additional data about titanium machinability, tool materials, coolant supply, shop talk, and even some historical anecdotes.

With compliments,

ISCAR team,
Your partner in intelligent machining



SYMBOLS AND UNITS*

The guide utilizes metric units and their equivalent US customary units as well.
As a rule, the latter are put in brackets and follow metric measurements.

AE	angle of engagement,
a	machining allowance (stock to be removed) per pass, mm (in)
ae	width (radial depth) of cut, mm (in)
ap	axial depth of cut, mm (in)
ap_{max}	maximum axial depth of cut, mm (in)
c	ratio of width of cut ae to nominal tool diameter d
d	nominal tool diameter, mm (in)
de	effective diameter, mm (in)
f	feed, mm/rev (IPR, ipr, inches per revolution)
fz	feed per tooth (chip load), mm/tooth (IPT, ipt, inches per tooth)
fzo	basic (initial) feed, (chip load), mm/tooth (IPT, ipt, inches per tooth)
fz_{min}	minimum feed per tooth (chip load), mm/tooth (IPT, ipt, inches per tooth)
fz_{max}	maximum feed per tooth (chip load), mm/tooth (IPT, ipt, inches per tooth)
fz_{av}	arithmetic average feed per tooth (chip load), mm/tooth (IPT, ipt, inches per tooth)
hm	average chip thickness, mm (in)
h_{max}	maximum chip thickness, mm (in)
Ke	engagement factor
Kf	tooth strength factor
Kh	overhang factor
Km	machinability factor
Ks	stability factor
n	rotational velocity (spindle speed), rpm (rpm)
Q	metal removal rate (MRR), cm^3/min (in^3/min)
Vo	basic cutting speed, m/min (SFM, sfm)
Vc	cutting speed, m/min (SFM, sfm)
Vf	feed speed (feed rate), mm/min (IPM, ipm, inch per minute)
V_{HP}	cutting speed in machining with high pressure coolant, m/min (SFM, sfm)
αr	ramping angle
χ	cutting edge (entering) angle

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

ABBREVIATION LIST

A	annealing (annealed)
ASTM	American Society for Testing and Materials
AM	additive manufacturing
BASCA	beta annealing followed by slow cooling and aging (beta annealed, slow cooled and aged)
Bling	bladed ring
Blisk	bladed disk
Blotor	bladed rotor
Blum	bladed drum
BUE	build-up edge
CAD/CAM	computer-aided design and manufacturing
CBN	cubic boron nitride
CNC	computer numerical control
CVD	chemical vapor deposition
DA	duplex annealing (duplex annealed)
Dia.	diameter
DOC	depth of cut
ELI	extra low interstitials
FF	fast feed*
HFM	high feed milling
HPC	high pressure coolant
HSM	high speed milling, high speed machining
HSS	high-speed steel
IBR	integrally bladed rotors (IBR)
ISO	International Organization for Standardization
MTB	machine tool builder (builders)
MRR	metal removal rate
PCD	polycrystalline diamond
PVD	physical vapor deposition
R	rolling (rolled)
SCEM	solid carbide endmills
STA	solution treatment and annealing (solution treated and annealed)
UHPC	ultra high pressure coolant
WOC	width of cut

***** fast feed is synonymic with high feed

TITANIUM GRADES

The common term “titanium” usually implies not only pure titanium, but also its alloys. For the most part, industry consumes just alloys. Alloying titanium is oriented to receiving grades with different properties and allows heat treatment for some of them. Typical alloying elements for titanium are Aluminum, Vanadium, Molybdenum, Ferrum, Chrome and others. Even small amounts of these elements can drastically change the properties of the alloys.

Pure titanium can exist in two crystalline forms: the low-temperature α-phase with hexagonal close-packed lattice and the higher-temperature β-phase with body-centered cubic lattice. The alloying elements in titanium alloys lead to an increase in α-phase or β-phase. Elements that strengthen, or stabilize a phase are called stabilizers: α-stabilizers or β-stabilizers. For example, Aluminum, Nitrogen and Plumbum are α-stabilizers, and Vanadium and Molybdenum are β-stabilizers. Some elements added to titanium alloys do not strengthen either the α- or β-phase, but are important as “neutral” elements that give certain properties to the alloy.

Based on its metallurgical characteristics, titanium is divided into the following groups:

- Commercially pure titanium (unalloyed), featuring a good corrosion resistance but low strength.
- α-alloys (alpha-alloys), consisting of only α-phase and having a lot of α-stabilizers, which feature strength retention at relatively high temperatures.
- Near-α-alloys (near-alpha-alloys), which are α-alloys with a small addition of β-stabilizers and have good resistance to creep for working temperatures 450°C -550°C.
- α-β-alloys (alpha-beta-alloys), perhaps the most common group, which is a type of mixture of both phases and contains α- and β-stabilizers. The alloys of this group are suitable for heat treatment and aging for strength improvement.
- β-alloys (beta-alloys) include a sufficiently large number of β-stabilizers to obtain β-phase structure after treatment or even cooling in some cases. The group is characterized by high hardenability and, therefore, high strength. However, increasing the amount of the alloying elements leads to higher density. In addition, increasing strength by solution treating and aging causes a reduction in ductility.

The family of near-β-alloys (near-beta-alloys) is sometimes separated from the group because it does not retain a fully β-phase structure after treatment.

Generally, the α-alloys have better ductility and the β-alloys have higher strength. The α-β-alloys lay between the mentioned two (**Fig. 1**).

There are different code systems for designating titanium. The American Society for Testing and Materials (ASTM), for instance, uses a grade numbering method: Grade 1, Grade 2, … , Grade 12,…

This guide utilizes the chemical composition system, where the designation shows the percentage of main alloying elements in an alloy. For example: “Ti6Al4V” or “Ti-6Al-4V” means that the titanium alloy contains 6% Aluminum (Al) and 4% Vanadium (V). The amount of the other alloying elements is small and, if necessary, can be found in detailed specification of the grade. The remaining component is titanium.

Table 1 shows some typical titanium grades and their use in industry.

Table 1 - Application of Selected Titanium Grades

Group	Designation	Conditions	Application examples
Commercially pure titanium	ASTM Grades 1-4	A	Medical equipment, implants, chemical equipment, marine and aircraft parts
α-titanium alloys	Ti-5Al-2.5Sn	A	Aircrafts (compressor blades, pipelines), steam turbine blades
near-α-titanium alloys	Ti-8Al-1Mo-1V Ti-6Al-1Mo-2Cb-1Ta	DA R	Airframes Welded corrosion-resistance parts
α-β-titanium alloys	Ti-6Al-4V Ti-7Al-4Mo	A; STA STA	Turbine and compressor blades, disks, fasteners, rocket parts Airframes, rocket parts
β- and near β-titanium alloys	Ti-13V-11Cr-3Al Ti-10V-2Fe-3Al Ti-5Al-5Mo-5V-1Cr-1Fe Ti-5Al-5Mo-5V-3Cr	A; STA STA A; STA STA	Fasteners, stressed structure elements, rocket parts High-strength parts, airframes, landing gears High-strength parts, airframes, landing gears High-strength parts, airframes, landing gears

A- annealed, DA - duplex annealed, R - rolled, STA - solution treated and aged

Titanium received its name from Martin Klaproth, a German chemist, one of the discoverers of titanium in the end of 18th century. Klaproth, who found the oxide of a new metal in an ore, gave it the name of the mythical giants – the Titans. However, there is another and more exotic version, according to which the source of the name "titanium" is Titania, the Queen of the Fairies and the wife of King Oberon in William Shakespeare's "A Midsummer Night's Dream" . With the combination of an ability to withstand titanic stress and its fairylike lightness, titanium fully justifies its name.

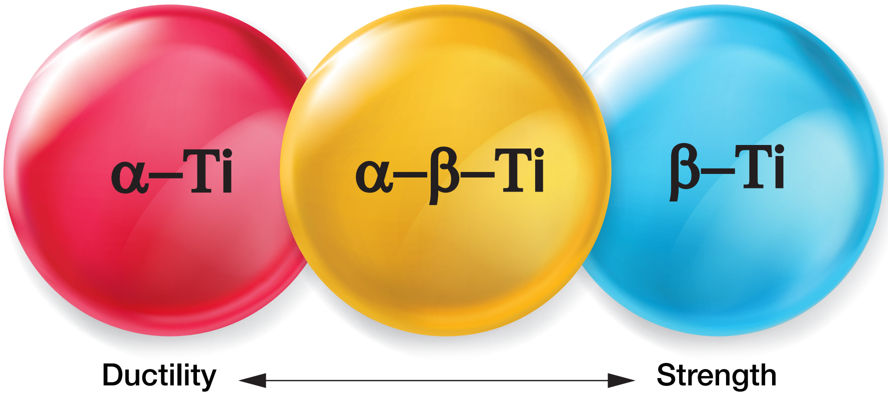


Fig. 1 Ductility-strength properties of titanium alloys

MACHINABILITY OF TITANIUM

There a widespread belief that titanium is like austenitic stainless steel in terms of its machinability. This may be true when relating to commercially pure titanium and also, with some assumption, α- or even α-β-alloys; however, it is fundamentally wrong with respect to the treated β- and near-β- alloys.

In general, titanium alloys (which we will refer to as titanium and specify their composition, grade and properties separately where necessary) are hard-to-machine materials and their machinability depends on various factors: chemical composition, hardness, method of treatment.
The main difficulties in cutting titanium are the following:

- Intensive heat generation leads to excessive adhesive wear of cutting edge.
- Low heat conductivity results in poor heat transfer and slowing heat dissipation down. Therefore, cutting edge experiences considerable thermal loading.
- “Springiness” of titanium due to low modulus of elasticity contributes to vibrations and worsens machining accuracy and surface finish.

The mentioned factors significantly reduce tool life and affect performance.
The averaged data in Table 2 allows estimating machinability of titanium compared with other groups of basic engineering materials.

Table 2 - Machinability of Titanium vs. Typical Engineering Materials (Averaged Data)

Material	ISO group	Machinability, %
Non-alloy free cutting steel	**P**	100
Low alloy steel, annealed		60
High alloy steel, annealed		50
Austenitic stainless steel, annealed	**M**	40
Commercially pure titanium	**S**	43
Titanium Ti-6Al-4V, annealed		25

At the same time, as already mentioned, titanium machinability varies depending on the titanium groups and the grades within the groups.
Table 3 provides a comparison of machinability for different titanium representatives; and **Fig. 2** shows an appropriate graphic expression.

Titanium is a difficult-to-cut-material and its machining is challenging.

Table 3 - Machinability Rating of Titanium Grades (Averaged Data)

Group	Designation	Condition*	Hardness	Machinability %
Pure titanium	Grade 2		HB 150-200	170
α- and near-α-titanium	Ti-5Al-2.5Sn	A	HRC 31-34	115
	Ti-8Al-1Mo-1V	DA	HRC 34-36	110
α-β-titanium	Ti-6Al-4V	A	HRC 32-36	100
	Ti-8V-5Fe-1Al	A	HRC 34-38	97
	Ti-6Al-4V	STA	HRC 39-41	90
β- and near-β-titanium	Ti-10V-2Fe-3Al	STA	HRC 35-42	56
	Ti-13V-11Cr-3Al	A	HRC 39-41	53
	Ti-5Al-5Mo-5V-3Cr	STA	HRC 36-44	51

* A - annealed, DA - duplex annealed, STA - solution treated and aged

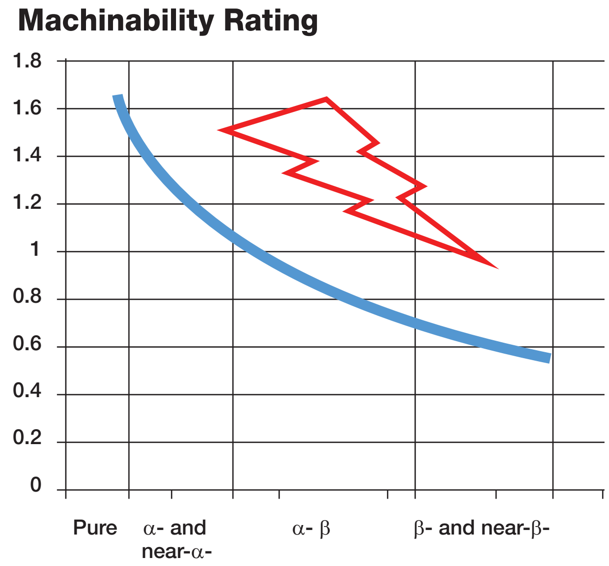


Fig. 2 Machinability of titanium groups (averaged data)

β-stabilizers, which contribute to higher strength, increase specific cutting force.
*As a rule, the **β**-phase structure reduces machinability and machining **β**-titanium is more difficult.*

Extensive use of titanium in industry, particularly in the aerospace sector, led to in-depth study of machining titanium and caused serious changes in technology. As a result, manufacturers built reliable metal cutting processes and took titanium machining productivity to a new level.
This reduced production costs and caused added momentum to a wide penetration of titanium in new designs. Today, for example, machining Ti-6Al-4V, the most commonly used titanium grade, is considered already as a closed chapter, although 15-20 years ago it was not a cakewalk.

Success has boosted interest in a wider application of other titanium grades featuring better strength but with poorer machinability. These grades focused the attention of designers as a good alternative to steel and stainless steel for weight saving but the low productivity presented a barrier. Following new demands, industry faced a problem with productive machining titanium alloys such as Ti-10V-2Fe-3Al and Ti-5Al-5Mo-5V-3Cr (VST 5553) that had a strong presence as key elements in different structural components. On the one hand, machine tool builders (MTB) achieve an impressive success in high-efficiency machine tools specifically dedicated to cutting titanium. These latest-generation multi-axis machines are intended for high volume production capability and feature high-torque main drive, advanced CNC software and adaptive control units, and high-pressure coolant supply. On the other hand, far slower progress in the cutting tools field has presented an obstacle in taking full advantage of the new machines.

Finding a way how to close the produced gap dictates strict requirements for cutting tools intended for machining titanium. The cutting tool, which often seems to be a minor link in a whole technological chain, actually plays a role of the prime factor, which can boost productivity of titanium and firstly its difficult-to-cut grades.

There are few options for substantial improvement of cutting tools. The main directions for the development of advanced tools relate to cutting tool materials and cutting tool design (geometry).

Cemented Carbide Grades for Milling Titanium

In indexable milling, ISCAR proposes several grades that were designed especially for machining ISO S materials including titanium. Some of the grades are the main grades; the others are considered as complementary. **Fig. 3** characterizes the application field of the grades in accordance with ISO 513.

> *Producers of titanium parts have high hopes for additive manufacturing (AM), that it will be a wind of serious changes in technology. AM allows forming a nearer-to-net shape part to minimize machining operations substantially. Removing high-volume material during rough cuts will be cancelled and it will automatically change the machine tool population of a company: heavy duty machines will simply disappear. However, despite the great success of AM, experts believe that introducing this innovative and promising method will take time. Strict safety requirements from the aerospace industry, the main consumer of titanium components, necessitate extreme caution in adoption of the new ideas.*

<table>
<tr><th colspan="9">ISO S GRADES - Indexable Milling</th></tr>
<tr><td>S05</td><td></td><td></td><td></td><td></td><td></td><td></td><td></td><td></td></tr>
<tr><td></td><td></td><td></td><td></td><td></td><td></td><td></td><td></td><td></td></tr>
<tr><td>S10</td><td></td><td rowspan="7">IC808
IC908</td><td></td><td></td><td></td><td></td><td></td><td></td></tr>
<tr><td></td><td></td><td></td><td></td><td></td><td></td><td></td><td></td></tr>
<tr><td>S15</td><td></td><td rowspan="5">IC840</td><td></td><td rowspan="3">IC380</td><td></td><td rowspan="5">IC5820</td><td rowspan="7">IC830
IC928</td></tr>
<tr><td></td><td></td><td></td><td></td></tr>
<tr><td>S20</td><td></td><td rowspan="5">IC882</td><td rowspan="5">IC330
IC328</td></tr>
<tr><td></td><td></td><td></td></tr>
<tr><td>S25</td><td></td><td></td></tr>
<tr><td></td><td></td><td></td><td></td><td></td><td></td></tr>
<tr><td>S30</td><td></td><td></td><td></td><td></td><td></td></tr>
</table>

▨ Main grade

□ Complementary grade

Post-coating SUMO TEC treatment

Post-coating treatment

IC380- Especially for milling Ti in stable conditions

Fig. 3 ISCAR carbide grades for indexable milling titanium

The main grades are represented by IC808/IC908 (the hardest grades), IC882 (the toughest grade) and IC840, which is in between.

The complementary grades demonstrate good results in specific applications, such as IC380 with its impressing performance when milling in stable conditions, and, therefore, they can be used in machining titanium as well.

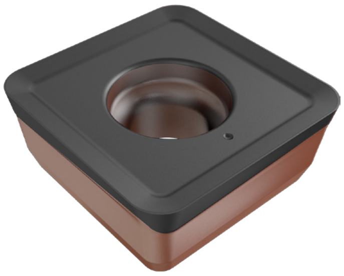


Fig. 4 Bronze “sun tan” carbide inserts

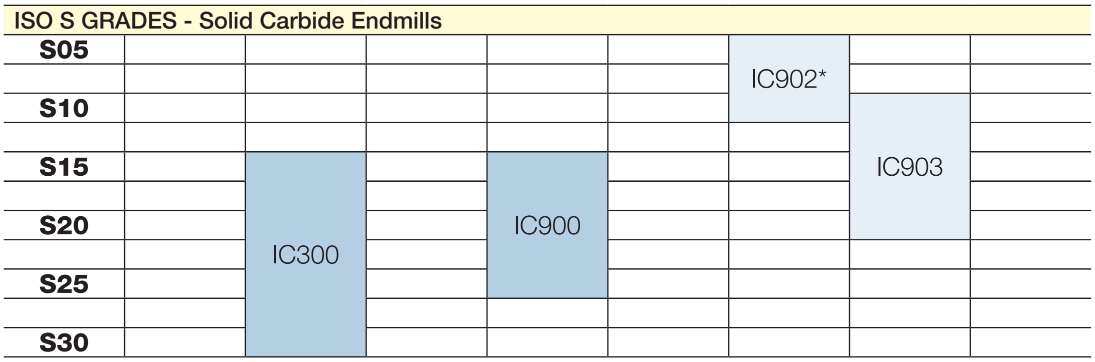
Some facts about carbide grades

A carbide grade is the combination of three elements: a tungsten carbide substrate, which is produced by powder-metallurgy technology, a wear-resistant coating and a post-coating treatment, while only although only the substrate is essential. In some cases a specific tungsten carbide may act as an uncoated grades; while in other cases it will be a substrate of a coated grade.

SUMO TEC is a special post coating treatment of indexable inserts, which provides substantially improved tool life and better reliability.
It has the effect of making the rake face of an insert even and uniform, minimizing inner stresses, cracks and droplets in coating that leads to smooth chip flow and extended tool life.

*ISCAR PVD coated grades IC840 and IC882 have a bronze chocolate color. The periphery of inserts, which are made from SUMO TEC CVD coated grade IC 5820, is also a brown shade. Development of these "sun tan" grades was directed exactly towards milling ISO S (titanium and heat-resistant alloys) and ISO M (austenitic and austenitic/ferritic stainless steel) materials. ISCAR believes that the above "sunbathed" cemented carbides (**Fig. 4**) will bring a real premium-class cutting "assorted chocolate" to the manufacturer of titanium parts.*

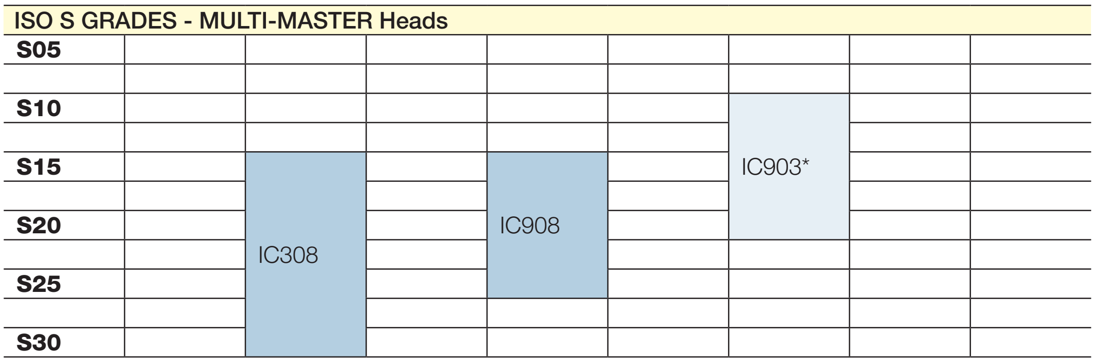
While the choice of carbide grades for solid carbide endmills (SCEM) and **MULTI-MASTER** end milling heads is somewhat limited, it is enough for producing a rich variety of milling tools that effectively cut titanium by different techniques, including a high speed machining (HSM) method. **Fig. 5** and **6** show the application field of the grades in concordance with ISO 513.



ISO S GRADES - Solid Carbide Endmills								
S05						IC902*		
S10							IC903	
S15		IC300		IC900				
S20								
S25								
S30								

▨ Main grade ░ Complementary grade * First choice for trochoidal milling

Fig. 5 ISCAR carbide grades for milling titanium by SCEM



ISO S GRADES - MULTI-MASTER Heads								
S05								
S10						IC903*		
S15		IC308		IC908				
S20								
S25								
S30								

▨ Main grade ░ Complementary grade * First choice for trochoidal milling

Fig. 6 ISCAR carbide grades for milling titanium by SCEM

How To Choose The Correct Carbide Grade?

As the people engaged in production like to say, the best carbide grade is the grade that you have in your stock. This statement (which can be related to the cutting tool as a whole) is probably true if it is a production situation that requires an operative decision. However, productive process planning, or effective tool stock managing requires more serious applicative analyzing of the pros and cons of the proposed carbide grades. **Fig. 3-6** with schematic representations of the application field of the grades, illustrate clearly the conclusions regarding recommended grade selection.

Indexable Milling Cutters

IC840 is the most universal carbide grade, and therefore it should be considered as the first-choice grade, especially in the most general cases of process engineering.

Grade IC808 features higher hardness compared to IC840. It is suitable mostly for semi-finish and finish operations and may be defined as a second-choice grade. IC882 is the strongest among the main cemented carbide grades for indexable milling titanium. It is intended primarily for heavy duty milling, machining in unfavorable conditions, for cases with significant impact load, etc.

With regards to the complementary grades, IC380 has proved itself, in milling when the rigidity of a technological system is high; and IC5820 shows impressive results in machining with high pressure coolant supply (HPC), preferably pin-pointed.

Comparative characteristics of grade performance in specific cases of indexable milling titanium are shown in Table 4.

Table 4 Performance of Carbide Grades In Specific Milling Applications

Grade	Comparative performance					
	Heavy duty Cutting	Fast feed Milling (FF)	HPC*	Unfavorable Conditions	High rigidity of system	Finishing at Small ae**
IC808	♦	◊	♦	♦	♦♦◊	♦♦♦
IC840	♦♦	♦♦	♦♦◊	♦♦	♦♦	♦♦
IC882	♦♦♦	♦♦♦	♦	♦♦♦	♦♦	♦◊
IC380	♦	♦	♦	♦	♦♦♦	♦♦
IC5820	♦◊	♦◊	♦♦♦	♦◊	♦♦◊	♦♦◊
IC830	♦◊	♦♦	♦	♦♦	♦♦	♦◊
IC330	♦♦	♦♦	♦◊	♦♦◊	♦◊	♦

* high pressure coolant supply
** ae - width of cut

Solid Carbide Endmills (SCEM) and MULTI-MASTER Milling Heads

The choice of carbide grades in SCEM and **MULTI-MASTER** families is a little lower compared to indexable milling cutters. The grades available are are more general and all-purpose from a practical standpoint, and feature a wide field of application. Table 5 shows the recommended carbide grades for typical and trochoidal milling.

Table 5 Choosing ISO S Carbide Grades for SCEM and MULTI-MASTER Heads

Grades	Typical milling techniques		Trochoidal milling	
	1st choice	2nd choice	**1st choice**	2nd choice
Solid carbide endmills	**IC300**	IC900	**IC902**	IC903
MULTI-MASTER heads	**IC308**	IC908	**IC903**	-

Example

The production engineering office of a manufacturer of titanium plant is planning comparative tests for indexable high feed milling cutters. The operation is milling the large-sized open plane surface of a titanium component. Technologists chose ISCAR **MILL4FEED** shell mills carrying inserts FFQ4 SOMT 120516HP. As the catalog shows, the inserts are available in grades IC882, IC830, IC5820 and IC808. The office decides to test two different grades. Which of them should be ordered?

High feed (fast feed in ISCAR terminology) milling causes considerable tooth load. In accordance with grade data (refer to **Fig. 3**), the available grades IC882, IC830, IC5820 and IC808 feature the following ISO application fields: (S20-S30), (S15-S30), (S15-S25) and (S10-S25) correspondingly. An increasing number after "S" characterizes growth of a grade toughness, grades IC882 and IC830 are more suitable for testing and therefore they should be ordered.

Example

Which carbide grade is the most effective for indexable milling with high pressure coolant supply? Table 4 provides a clear answer to this question: IC5820.

Nano layered PVD coating

PVD coatings were introduced during the late 1980's. With the use of advanced nanotechnology, PVD coatings performed a gigantic step in overcoming complex problems that were impeding progress in the field.

Developments in science and technology brought a new class of wear-resistant nano layered coatings. These coatings are a combination of layers having a thickness of up to 50 nm (nanometers) and demonstrate significant increases in the strength of the coating compared to conventional methods.

General Guidelines for Milling Titanium

When milling titanium, a manufacturer tries to obtain the most effective milling technique and the appropriate tool. Best production practices have already developed general recommendations to successfully complete these tasks, in order to overcome the main difficulties in cutting titanium while ensuring acceptable productivity and tool life. Even though the technique and the tool relate to each other, the recommendation may be considered with respect to each separately.

Milling Technique

Milling technique or milling strategy determines the tool path and the "depth of cut (ap) – width of cut (ae)" relation. In choosing the most suitable machining strategy, the following points are taken into consideration:

A rotating mill contacts a machined workpiece by arc that is measured by angle of engagement AE (**Fig. 7**). Decreasing this arc (i.e. width of cut ae) reduces the heat load on a cutting edge of the mill. In addition, it increases the interval during which the edge is involved directly in cutting, and so ensures more time for edge cooling. Less heat generation diminishes the risk of titanium work hardening during machining.
Due to the above factors, reducing ae allows increasing cutting speed Vc. In milling full slot directly from solid with cutting speed Vc1, the width of cut is equal to tool diameter d. In comparison with this case, in milling square shoulder with ae less than 0.1×d (AE≈37°) the cutting speed may be increased by 150-200% (1.5…2×Vc1). **Fig. 8** shows an approximate plot of Vc against AE and ae in milling slot in a workpiece from Ti-6Al-4V by different methods.

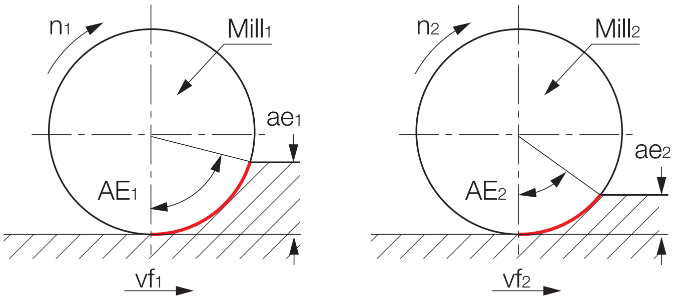


Fig. 7 Contact arc and angle of engagement

3. An approach cut by arc ("rolling in") is preferable (**Fig. 9**). When a milling cutter enters a machined material by arc, the mechanical and the thermal loads on the cutting edge grow gradually and not suddenly, which significantly contributes to machining stability and improving tool life.
4. Today with the use of advanced CAD/CAM systems it is possible to plan a tool path with a practically constant angle of engagement. This can constrain the arc of contact to prevent both overloading and overheating the tool.
5. In cutting, when the temperature in a cutting zone is high, a chemical interaction between the cutting edge and the material, as well as edge oxidation, results in notch wear. If a milling cutter machines a deep square shoulder by passes with constant depth of cut ap per pass, a notch is more likely to occur. This notch causes deformation in the material instead of cutting it, which leads to material work-hardening and scoring the material surface, resulting in abnormal cutting conditions and poor surface finish. Therefore, varying ap per pass in multi-pass milling reduces intensive notch wearing in located area and diminishes these negative effects.

6. Titanium alloys, especially their more difficult-to-machine grades, feature significant specific cutting force, which leads to high mechanical loading of the cutting edge. Titanium's "springiness" boosts vibrations, particularly in rough milling with considerable machining allowance. This means that effective cutting under such conditions stipulates high rigidity of the whole technological system, and stiffness of a machine tool, proper work- and tool holding, and the tool overhang may be crucial.
7. In process planning, check the possibility of milling titanium while it is in a soft condition. Heat treatment and age-hardening make titanium much more difficult to machine.
8. High-pressure coolant (HPC) supply can dramatically improve milling efficiency*.
9. Milling deep pockets and cavities requires a high tool overhang. When planning machining operations, possible alternatives are to use several milling cutters with different overhangs, or several assemblies, in which the same cutters are mounted in toolholders with various neck lengths, instead of one long-reach tool.
10. High feed milling (HFM, fast feed milling) may be a viable alternative to traditional heavy-duty rough milling. HFM titanium is suitable for machining tools with slow table drive due to the feed speed being considerably lower compared to the values necessary for HFM steel. However, applying the alternative requires machines with a rigid spindle unit.

Trochoidal milling

Trochoidal milling is a high efficiency milling technique that uses advantages of decreasing the angle of engagement, machining with thin chip, and approach cut by arc, which result in increasing cutting speed and feed rate. In trochoidal milling, a fast-rotating tool that operates at high depth of cut, moves along arc and "slices" a thin but wide layer of material. When the layer is removed, the cutter advances deeper into the material radially and then repeats the slicing. This method ensures uniform tool engagement and stable average chip thickness. Therefore, the tool experiences constant load that causes uniform wear and predictable tool life. The small thickness of sliced material both significantly reduces heat impact on the tool and ensures increasing the number of the tool teeth. As a result, the method ensures very high metal removal rate with considerably decreased power consumption and improved tool life. Trochoidal milling is very popular in producing titanium parts, for example in rough and semi-finish machining slots in blisks (bladed disks) and impellers.

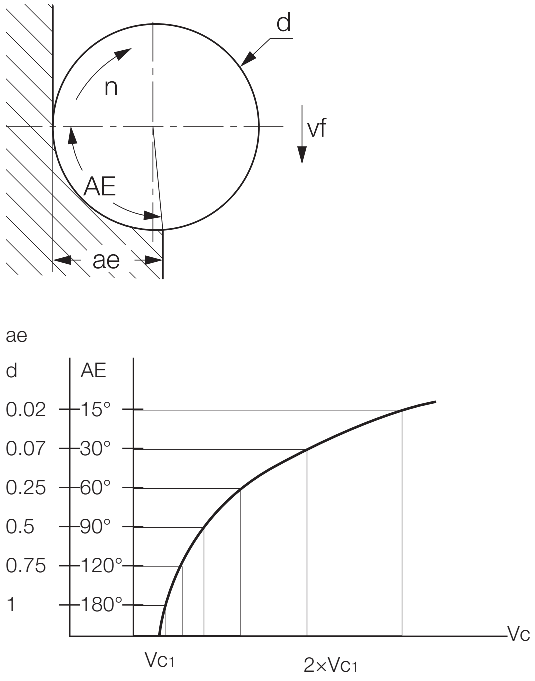


Fig. 8 Plot Vc against AE and ae

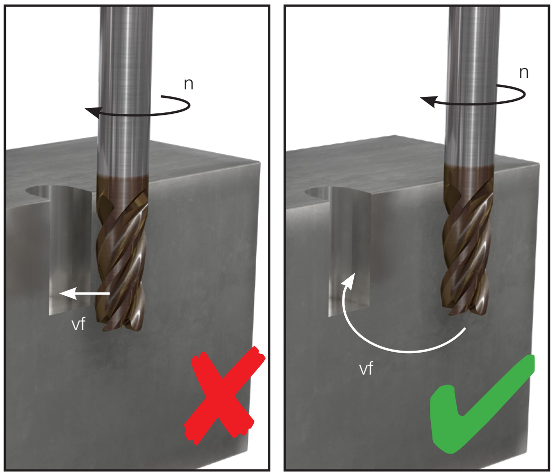


Fig. 9 Direct entering into material and entering by arc ("rolling in")

---

* HPC benefits are considered in more detail in the following pages of the guide

Milling Tool

Selecting a suitable tool for milling titanium is a very important issue. The following three features: cutting geometry, tool material, and design configuration, should be examined.

1. It is considered that tools for milling titanium feature positive-rake geometry with appropriate edge preparation. This is quite right with respect to commercially pure and easy-to-machine titanium grades. However, milling the difficult-to-machine grades requires a reinforced cutting edge that is ensured by neutral or negative T-land. In many cases of roughing applications (such as heavy-duty milling by indexable extended flute milling cutters or HFM), tools with negative-rake geometry, which provide higher feed, are the best choice.
2. Milling titanium dictates increasing relief angle.
3. When indexable milling tools are examined, cutters with one-sided inserts are preferable. Generally, they demonstrate better performance when compared to tools carrying double-sided inserts. However, in rough machining wide edges of workpieces (edging), the indexable extended flute cutters with tangentially clamped double-sided inserts can be a first-choice solution.
4. The tool material and design should ensure a sharp edge during reasonable tool life. In conditions of intensive heat generation, a dull ("bad") edge increases build-up edge formation and tool wear, and leads to serious tool damage.
5. Machining titanium features considerable heat build-up and mechanical load, particularly for difficult to machine grades, and the tool material must maintain a correct balance between high-temperature strength and impact resistance.
6. Coated cemented carbides are the main tool materials for titanium milling. PVD coating retains a sharp cutting edge better than CVD. The majority of indexable carbide inserts intended for milling titanium are PVD-coated. However, CVD-coated inserts, which are usually less sharp than PVD-coated inserts, often perform better in heavy-duty rough milling.
7. Cutting ceramics and cubic boron nitride (CBN) are not so suitable for machining titanium, although polycrystalline diamond (PCD) has proved itself in finish milling in several cases.
8. Most machine tools today are usually designed to supply coolant through a spindle. Therefore, applying cutting tools with internal coolant supply through the tool body is preferable.
9. Vibration strength in end milling can be improved drastically by using solid carbide endmills (SCEM) with chatter-free cutting geometry.
10. Milling cutter accuracy is an important factor not only for ensuring the required precision parameters in finishing operations. Low accuracy, especially in case of small-diameter cutters, affects performance and reduces tool life. For example, excessive run out leads to increased loading of a tooth and its early wear, which causes overloading the other teeth and even cutter failure. Poor cylindricity of the tool shank diminishes the effective area of contact between the shank and a toolholder that significantly decreases clamping force.

Initial Cutting Data In Milling

Generally, cutting data relates to finding depth and width of cut (ap and ae correspondingly), cutting speed Vc and feed per tooth fz. Defining ap and ae is strongly dependent not only on a chosen milling method and process planning but also on tool design characteristics such as diameter and maximal cutting length. This section of the guide considers Vc and fz estimation.

Cutting Speed

Vc varies within a broad range, depending on the type of machining and machinability of a specific titanium grade. Typical Vc values are shown in Table 6.

Table 6 Typical Cutting Speed Vc When Milling Titanium

Type of Machining	Examples	Cutting speed Vc, m/min (sfm)
Hard (H)	Heavy-duty roughing, milling in unfavorable conditions	25-60 (80-200)
Medium (M)	Medium-duty roughing, semifinish milling	40-80 (130-260)
Light (L)	Finish milling, light-duty cutting	70-120 (230-400)

A more accurate estimation of starting speed Vc requires a simple calculation in accordance with equation (1).

$$Vc=Vo\times Km\times Ke\times Ks \qquad (1)$$

Where: ***Vo*** – basic cutting speed
Km – machinability factor
Ke – engagement factor
Ks – stability factor

Basic Cutting Speed Vo

Basic cutting speed Vo depends on a tool material. Table 7 gives Vo values for the carbide grades that are used in indexable milling cutters, and Table 8 gives Vo values for the carbide grades intended for SCEM and **MULTI-MASTER** heads.

Machinability Factor Km

Machinability factor Km reflects the difference in machinability for titanium grades. It is a derived value of machinability rating (Table 3, **Fig. 2**). Km is shown in Table 9.

Engagement Factor Ke

Reducing the angle of engagement AE (i.e. width of cut ae) allows increasing the cutting speed Vc (**Fig. 8**). Table 10 gives appropriate values Ke for correction.

Stability Factor Ks

Stability factor Ks is defined by the estimate below of milling operation stability:

- Ks=1 for normal stability
- Ks=0.7-0.8 for unstable operations (high overhang, poor tool- or workholding, milling thin walls, etc.) and milling in unfavorable conditions.

Cutting in unfavorable or unstable conditions? Heavy or heavy-duty machining?

"Cutting in unfavorable or unstable conditions" – which is correct? The meaning of these two definitions is confusing sometimes. "Unfavorable conditions" relate to the following cases of cutting: skinned workpiece, variable machining allowance (material to be removed) that leads to changing the depth of cut, significant impact load, surfaces with high-abrasive inclusions, chatter. "Unstable conditions" characterize low stability of a complete system (machine tool, workpiece holding fixture, cutting tool, workpiece) caused by poor tool and/or workpiece holding, non-rigid machine tool, high tool overhang, thin-walled workpiece, etc. Despite the gap in definitions, the conditions are in a cause-and-effect relationship: for example, low technological rigidity contributes to increased vibrations and thus worsens cutting conditions. Therefore, in some instances the above definitions are considered as synonyms.

Regarding heavy and heavy-duty machining. "Heavy-duty" means significant mechanical and thermal loading of the technological system and its separate components, including the cutting tool, due to removing considerable allowance, extreme cutting data, etc. "Heavy" is used with respect to machining large-sized heavy-weight parts on powerful heavy machine tools. So, "heavy-duty" specifies the mode of machining, a degree of loading the tool; and "heavy" is used more to define dimensions of the machined parts and metal cutting tools. Note that a small-size cutting tool can work in heavy-duty cycles under extremely high load and that producing large parts may require applying the small tool as well.

A "golden rule" for manufacturer engineer, process planner and machinist says: "Avoid heavy-duty machining in unfavorable conditions especially if your technological system is unstable!" This holds for all metalworking branches and among them heavy industry.

Table 7 Basic Cutting Speed Vo for Indexable Milling Cutters

Type of Machining	Vo for Carbide Grade of Inserts, m/min (sfm)						
	Main Grades			Complementary Grades			
	IC808 / IC908	IC840	IC882	IC5820	IC380	IC830 / IC928	IC330 / IC328
Hard (H)	43 (141)	43 (141)	38 (125)	43 (141)	40 (131)	40 (131)	38 (125)
Medium (M)	53 (174)	48 (157)	43 (141)	48 (157)	45 (147)	45 (147)	43 (141)
Light (L)	65 (213)	60 (197)	50 (164)	60 (197)	55 (180)	55 (180)	50 (164)

Table 8 Basic Cutting Speed Vo for SCEM and MULTI-MASTER Heads

Type of Machining	Vo for Carbide Grades, m/min (sfm)			
	Main Grades		Complementary Grades	
	IC900 / IC908	IC300 / IC308	IC902*	IC903*
Hard (H)	43 (141)	40 (131)	-	-
Medium (M)	53 (174)	45 (147)	65 (213)	58 (190)
Light (L)	65 (213)	55 (180)	75 (246)	70 (230)

* The grades are recommended mainly for HSM by trochoidal method

Table 9 Machinability Factor Km for Titanium Grades

Group	Typical representative			Machinability Factor Km
	Designation	Condition*	Hardness	
Pure Titanium	Grade 2		HB 150-200	1.70
A-and near-α Titanium	Ti-5Al-2.5Sn	ELI	HRC 31-34	1.25
	Ti-5Al-2.5Sn	A	HRC 31-34	1.15
	Ti-8Al-1Mo-1V	DA	HRC 34-36	1.10
A-β Titanium	Ti-6Al-4V	ELI	HRC 30-35	1.10
	Ti-6Al-4v	A	HRC 32-36	1.00
	Ti-8V-5Fe-1Al	A	HRC 34-38	0.97
	Ti-6Al-4V	STA	HRC 39-41	0.90
B-and near-β Titanium	Ti-5Al-5Mo-5V3Cr	BASCA	HRC 34-36	0.57
	Ti-10V-2Fe-4Al	STA	HRC 35-42	0.56
	Ti-13V-11Cr-3Al	A	HRC 39-41	0.53
	Ti-5Al-5Mo-5V-3Cr	STA	HRC 36-44	0.51

* A- annealed, BASCA- beta annealed, slow cooled and aged, DA - duplex annealed, ELI - extra low interstitials, STA - solution treated and aged

Table 10 Engagement Factor Ke

Ae/d	1.00	0.93	0.75	0.5	0.37	0.25	0.12	0.10	0.07
AE	180°	150°	120°	90°	75°	60°	40.5°	36°	30°
Ke	0.80	0.90	0.95	1.00	1.05	1.20	1.30	1.50	1.80

Example

Find the cutting speed for a face milling operation performed by a **DOVEIQMILL** indexable milling cutter carrying inserts, made from carbide grade IC840.
The cutter diameter is 100 mm (4 in), the set width of cut is 70 mm (2.75 in) and the width of cut 2.5 mm (.1 in). The workpiece material is Ti-6Al-4V, annealed.
Assume that operation stability is sufficient (face milling by the cutter of relatively big diameter, short overhang, good conditions of a machine tool, proper workpiece clamping) and machining type is medium (ap is 2.5 mm).

1. Rough estimating with the use of Table 6: starting speed Vc=60 m/min (197 sfm)
2. More accurate calculations using equation (1):
   Vo=48 m/min (157 sfm), Km=1, Ke≈0.96 (Tables 7, 9, 10 correspondingly),
   Ks=1.Vc=48×1×0.96×1=46 m/min (151 sfm)

The discrepancies in the received results are significant: rough estimating results in 30% more cutting speed.

Example

During process planning, a technologist examines applying an 80 mm (3 in) Dia. indexable extended flute shell mill for rough machining deep shoulder. The workpiece material is “triple 5 titanium” (Ti-5Al-5Mo-5V-3Cr), HRC 39-41. Cutting parameters: ap=85 mm (3.35 in), ae=20 mm (.79 in), wet coolant supply – internal. The carbide grade of inserts mounted in the shell mill is IC882. The operation will be performed on a new machine tool, workholding is rigid and, despite a relatively high overhang of the shell mill, which is mounted on arbor, operational stability is normal. Which Vc should be chosen for further calculation and analysis? Assuming that machining is hard, with the use of Tables 7, 9 and 10:
Vo=38 m/min (125 sfm), Km=0.51, Ke=1.2 (ae/d=0.25). Ks=1.
Vc=38×0.51×1.2×1=23 m/min (76 sfm)
Rough estimating with the use of Table 6 (the lower speed limit for heavy roughing) gives 25 m/min (80 sfm).

Poor machinability of hard-to cut titanium grades is a serious obstacle for increasing productivity in rough milling. To overcome this problem, industry uses progressive machining methods such as high speed milling (HSM), high feed milling (HFM) and machining with high pressure coolant (HPC) supply.

Example

In semi finish trochoidal milling slots in blisk (bladed disc) produced from
annealed Ti-6Al-4V, a 7-flute 10 mm (.4 in) Dia. solid carbide endmill
(SCEM) is applied with the following cutting parameters:
ap=40 mm (1.57 in), ae=0.8 mm (.031 in).
The SCEM material is carbide grade IC902. Estimate starting Vc.

The operation can be considered as light-duty cutting.
In accordance with data in Tables 8-10:
Vo=75 m/min (246 sfm), Km=1, Ke≈1.8 (ae/d=0.08).
Ks=1. Vc=75×1×1.8×1=135 m/min (443 sfm)

Feed Per Tooth

Cutting speed Vc, feed per tooth (advance per tooth) fz is a key parameter in milling. In the USA, it is often called “chip load” – fz reflects largely mechanical and thermal load on the tooth of a milling cutter. Increasing fz thickens generated chips that affects the ability of the chips to remove heat from the cutting zone, and causes a growth of cutting forces. However, seriously reduced fz results in chips that are too thin, worsens cutting action and dramatically reduces tool life. Therefore, selecting the feed per tooth should ensure an appropriate chip thickness, and maintaining the same chip thickness during a cutting process is a formula of good performance. While advanced CAD/CAM systems can program a tool path that maintains constant chip thickness, operating via feed per tooth is easier to understand and more practical, so metal cutting workers require this information.

In milling, the chip has complicated variable-thickness shape, while the thickness changes from minimum to maximum. Average chip thickness hm is used for cutting characteristics, engineering calculations and selecting fz. There is a mathematical relationship between average chip thickness, feed per tooth, width of cut and tool diameter - hm may be considered as a function of fz and vice versa. The average chip thickness is a computed value, but this feature does not diminish its importance: hm characterizes mechanical load on a milling cutter and a machine tool; in experienced hands it is one of the key parameters for estimating milling performance.

Various sources of technical information recommend the following equation for approximate calculating hm if the cutting edge angle of a milling cutter is 90°:

$$hm=fz\times \sqrt{ae/d} \qquad (2)$$

$$fz=hm\times \sqrt{d/ae} \qquad (2a)$$

Equation 2 is simple and easy to use in shop floor conditions, and gives good results that are suitable for finding hm for peripheral milling (machining by cylindrical or slab mill, 90° endmill – **Fig. 11**) and one kind of 90° face milling (**Fig. 12**). Also, it may be accepted for milling slots and grooves by a disc milling cutter in some cases. In other circumstances, the equations for average chip thickness differ. Tables 11 and 12 provide summarized data for finding hm and maximum chip thickness hmax that are suitable for rapid calculations under factory conditions.

*There are different methods for calculating average chip thickness in milling hm such as arithmetic mean, weighted mean, or a result based on finding the cross-sectional area of an assumingly undeformed chip and its substitution by simple rectangular, but the most common method is to compute average chip thickness in relation to half of an angle of engagement AE (**Fig. 10**).*

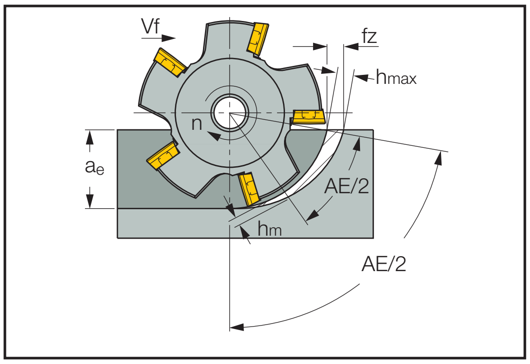


Fig. 10 Average (hm) and maximum (hmax) chip thickness in milling

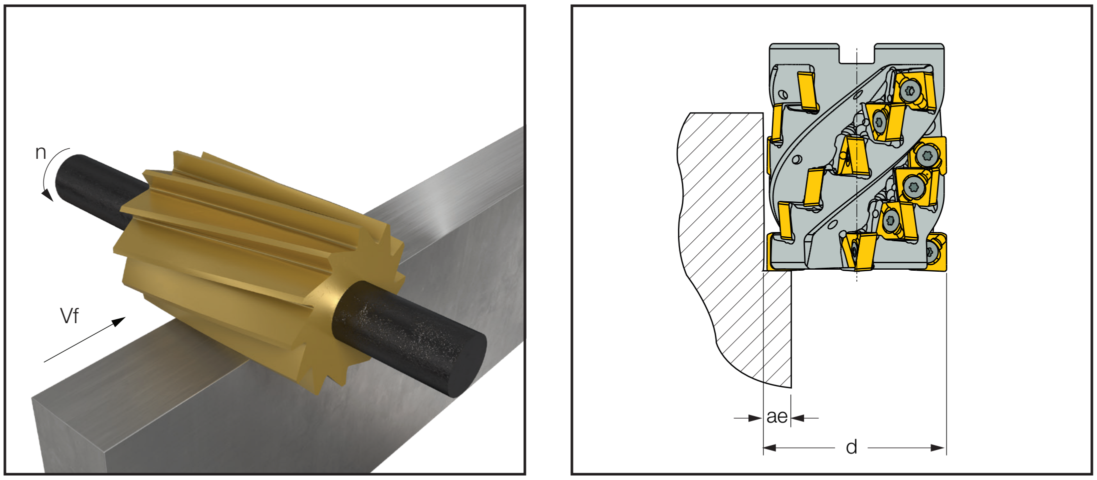


Fig. 11 Slab milling and milling deep shoulder

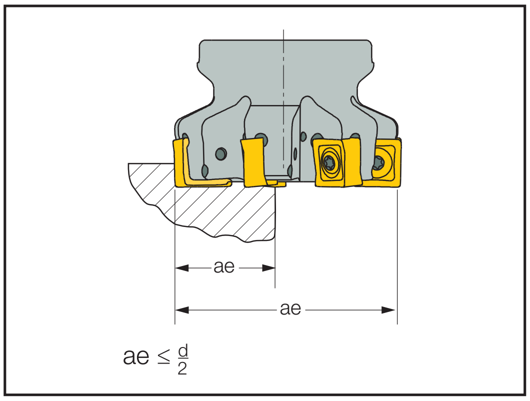
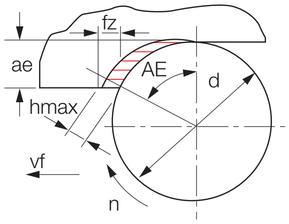
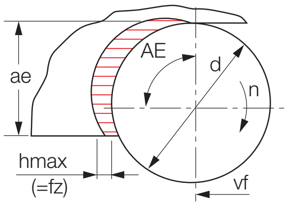


Fig. 12 Face milling with depth of cut no more than the radius of a milling cutter

Table 11 Calculating Chip Thickness In Peripheral Milling

Condition	Figure		Hm, hmax
	Example	Cutter position	
AE≤90°* (ae≤d/2)	**Fig. 10**		$hm=fz\times\sqrt{ae/d}$ $hmax=fz\times\sin AE$ $hmax=2\times fz\times\sqrt{ae\times d-ae^2}/d$
AE>90°* (ae>d/2)			<u>**1st method**</u> $hm=fz\times\sin(AE/2)$ $hmax=fz$ <u>**2nd method**</u> $hm=fz\times[\sqrt{2}/2+\cos((AE-90°)/2)]$ $hmax=fz$

* also suitable for milling slots by disc milling cutters

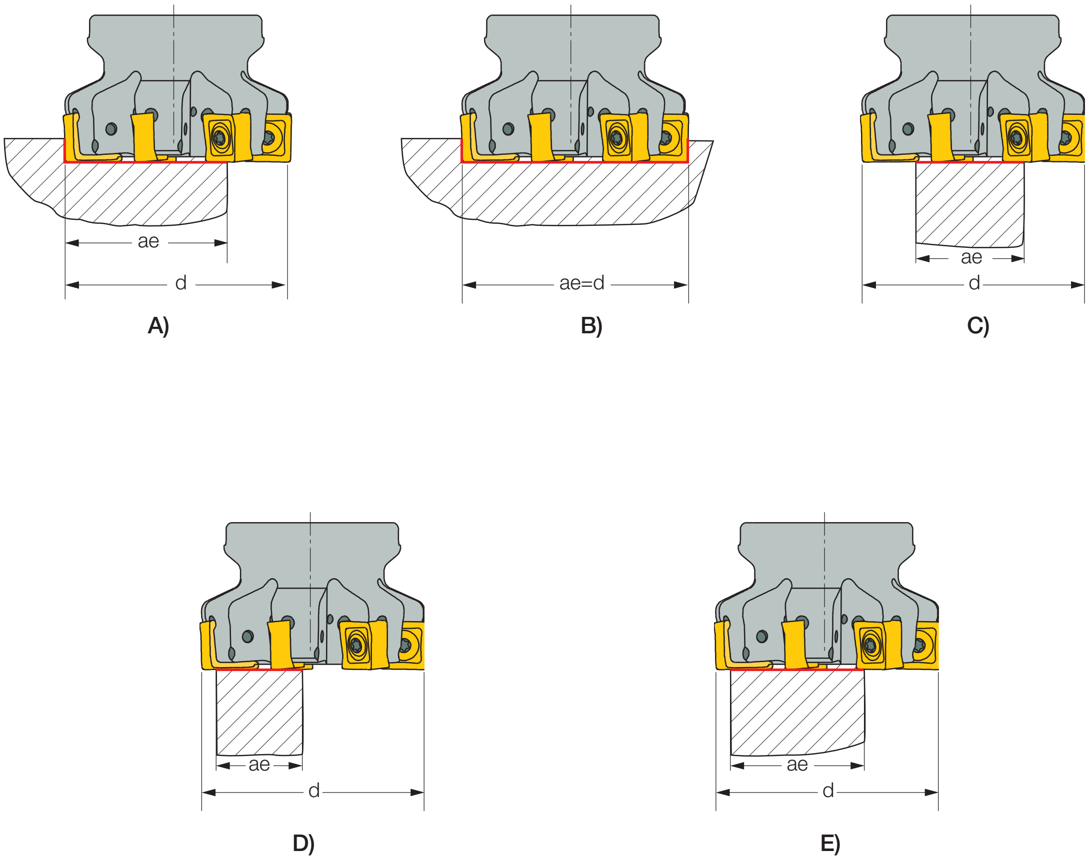
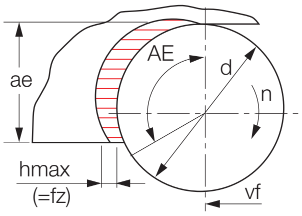
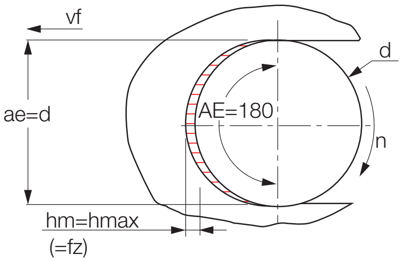
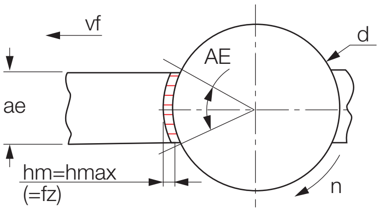
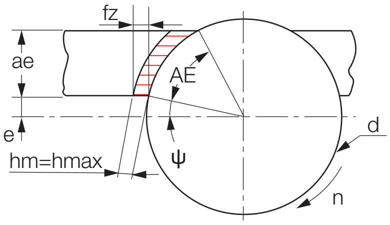
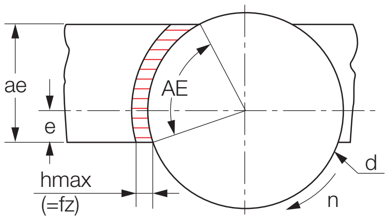


Fig. 13 Possible application of 90° face mills

Table 12 Calculating Chip Thickness In Face Milling

Example	Cutter position	Hm, hmax
Fig. 13 Case a)		**1**st method $hm=fz\times\sin(AE/2)$ $hmax=fz$ 2nd method $hm=fz\ 2\times(\sqrt{2}/2+\cos(AE-90^\circ)/2)$ $hmax=fz$
Fig. 13 Case b)		$hm=hmax=fz$
Fig. 13 Case c)*		$hm=hmax=fz$
Fig. 13 Case d)		$hm=fz\times\cos(\Psi+AE/2)$ $hmax=fz\times\cos\Psi$
Fig. 13 Case e)		1st method $hm=fz\times\sin(AE/2)$ $hmax=fz$ 2nd method $hm=hmax=fz$

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

Example

According to a manufacturing method, a 90° face mill of 100 mm (4 in) diameter is planned to machine the open plane surface of a part. The surface width is 75 mm (3 in). What is the value of feed per tooth that should be set to ensure average chip thickness 0.1 mm (.004")?

Referring to Table 12 and Fig. 13, case a), $hm=fz\times\sin(AE/2)$ $fz=hm/\sin(AE/2)$.
Angle of engagement $AE=90°+\arcsin((ae-r)/r)$ where $r=d/2$ – the radius of the face mill.
$AE=90°+\arcsin((75-50)/50)=120°$, $AE/2=60°$ and $fz=0.1/\sin 60°=0.11$ (mm/tooth).

For a 4 in. dia. cutter: $AE=90°+\arcsin((3-2)/2)=120°$, $fz=.004/\sin 60°=.0046$ (ipt).

Example

A deep square shoulder in a titanium part is roughly machined by an 80 mm (3 in) diameter indexable extended flute cutter that is operated with the following data:
$ae=20$ mm (.75 in),
$fz=0.2$ mm/tooth (.008 ipt).
Find the chip parameters.

With the use of Table 11 and **Fig. 11**:
- average chip thickness $hm=fz\times\sqrt{(ae/d)}=0.2\times\sqrt{(20/80)}=0.1$ (mm),
- maximum chip thickness
$hmax=2\times fz\times\sqrt{(d\times ae-ae^2)}/d=2\times0.2\times\sqrt{(80\times20-20^2)}/80=0.17$ (mm).

Accordingly, for the inch-size cutter:
$hm=.008\times\sqrt{(.75/3)}=.004$ (in),- $hmax=2\times.008\times\sqrt{(3\times.75-.75^2)}/3=.007$ (in).

Radial chip thinning
The given examples show that the calculated values of both average hm and maximum chip thickness hmax are lower than feed per tooth (chip load) fz. The examples illustrate one conclusion that can be made when examining Tables 11 and 12: if width (radial depth) of cut ae in peripheral milling and face milling (cases a) and d)) is less than the radius of a milling cutter, hmax becomes lower than fz. Reducing ae leads to decreasing hmax and, accordingly, hm. This effect is known as "radial chip thinning", and taking it into account is very important, especially in milling titanium which features intensive heat generation. Produced chips transfer most of the heat and, therefore, they should be thick and massive enough to retain the heat and carry it away. Maintaining the feed per tooth should ensure the required chip thickness and avoid critical chip thinning. Understanding this effect is a key element for correctly programmed fz.

Tables 13 and 14 give auxiliary data that can help in quickly calculating chip thickness for more common cases of milling:
- Table 13 for peripheral milling when AE<90° (or ae<d/2),
- Table 14 for face and peripheral milling when AE>90° (ae>d/2).

The following formulas may be useful for estimating AE in peripheral milling with ae<d/2:

$$AE=\arcsin(2\times\sqrt{c-c^2}) \qquad (3)$$

and

$$AE=\arccos(1-2\times c) \qquad (3a)$$

where $c=ae/d$ – ratio of width of cut ae to nominal tool diameter d.

Table 13 Auxiliary Data for Peripheral Milling When AE<90°

Ae/d	0.05	0.1	0.15	0.2	0.25	0.3	0.35	0.4	0.45
$\sqrt{ae/d}$	0.224	0.316	0.387	0.447	0.5	0.547	0.591	0.632	0.67
SinAE	0.436	0.6	0.714	0.8	0.866	0.916	0.954	0.98	0.995
AE	25.8°	36°	45.6°	53°	60°	66.4°	72.5°	78.5°	84.3°

Table 14 Auxiliary Data for Face and Peripheral Milling When AE<90°

Ae/d	0.55	0.65	0.75	0.85	0.95
AE	95.7°	107.5°	120°	134.4°	154°
AE/2	47.8°	53.7°	60°	67.2°	77°
Sin (AE/2)	0.741	0.806	0.866	0.921	0.974

Example

Find chip thickness for a deep square shoulder in a titanium part is roughly machined by an 80 mm (3 in) diameter indexable extended flute cutter that is operated with the following data:
- width of cut ae=60 mm (2.25 in),
- feed fz=0.15 mm/tooth (.006 ipt).

Basing on **Fig. 13**, case a), and Table 12, average chip thickness $hm=fz\times\sin(AE/2)$ and $hmax=fz$.
Due to $ae/d=60/80=0.75$, from Table 14 $AE=120°$ and $\sin(AE/2)=0.866$.
So, $hm=0.15\times0.866=0.13$ (mm) and $hmax=0.15$ mm.

Note. Estimating hm with the use of 2nd method (Table 12) gives the result as below:
$hm=fz/2\times[\sqrt{2}/2+\cos((AE-90°)/2)]$
$=0.15/2\times(\sqrt{2}/2+\cos15°)=0.125$ (mm)

Similarly, for 3 in. Dia. milling cutter:
$ae/d=2.25/3=0.75$, $\sin(AE/2)=0.866$, $hm=.006\times0.866=.005$ (in), $hmax=.006$ in.

Note. Estimating in accordance with the 2nd method (Table 12) will give
$hm=.006/2\times(\sqrt{2}/2+\cos15°)=.005$ (in)

Unfavorable, unstable, heavy and heavy-duty examples
The above terms may be simply explained with everyday life analogies:

Imagine that you are driving along a road. If your road is horseshoe and undivided, covered with stones or pitted – you are driving in unfavorable conditions.

If you are driving in a car carrying two bicycles on an upright mount on the car roof – you are driving in unstable conditions.

A large-size truck transporting a caterpillar excavator is a typical heavy vehicle.

If the driver of a light pickup car, which is loaded to maximal capacity, decides to overtake the truck, the pickup car motor will work in heavy-duty conditions.

☛ *If the cutting edge angle of a milling tool is not 90° (**Fig. 14**), the equations (2) and (2a) change to*

$$hm=fz\times\sqrt{(ae/d)}\times\sin\chi \quad (4)$$

and

$$fz=hm\times\sqrt{(d/ae)}\times1/\sin\chi \quad (4a)$$

correspondingly

Here χ – cutting edge (entering) angle

In the same manner, the cutting edge angle should be taken into account in calculating hm and hmax with the use of the equations in Tables 11 and 12.

Example

A planner of the process department that works under improving technology, examines applying face mill IQ845 FSY D125-09-40-R07 carrying inserts IQ845 SYHU 0704ADN-MM 808 to machining the top surface of a titanium component.
The catalog specifies the insert feed range as 0.15-0.3 mm/tooth (.006-.012 ipt) and the cutting tool angle of the mill as 50°. The mill diameter is 125 mm (4.92 in).
At which feed should the mill be operated to ensure average chip thickness 0.12-0.15 mm (.005-.006 ipt), if the depth of cut and the width of cut are 3.5 and 100 mm correspondingly?

With a use of equation (4a) the feed per tooth for ae=100 mm (4 in) will be $0.12\times\sqrt{125/100}\times 1/\sin 50^\circ = 0.18$ (mm/tooth) (.007 ipt) and $0.15\times\sqrt{125/100}\times 1/\sin 50^\circ = 0.22$ (mm/tooth) (.0087 ipt) for average chip thickness 0.12 and 0.15 mm correspondingly.

Hence, the programmed feed can be recommended of 0.2 mm/tooth (.008 ipt) with a possible increase of up to 0.22 mm/tooth (.0087 ipt) for maximal productivity.

Note. In brief calculations that are made without taking chip thickness as an important factor in setting correct cutting data, the feed per tooth is found in the following manner:
the required hm is (0.12+0.15)/2=0.135 (mm) (.0053 in) in average,
if consider the found value as hmax,
fz=hmax/sin 50°=0.135/ sin 50°=0.176≈0.18 (mm/tooth) (.007 ipt).
Alternatively, fz can be calculated, if 0.15 mm will be referred as hmax:
fz=0.15/sin 50°=0.195≈0.2 (mm/tooth) (.008 ipt).
Comparing these results with the values found previously shows that considering chip thickness as a function of ae gives more precise data and ensures better productivity.
Another simplified way for setting the feed is simply to take the average value from a declared feed range as the programmed data. In our example it will be
(0.15+0.3)/2=0.22 (mm/tooth) (.0087 ipt). In our case it is equal to fz, which was found for hm=0.15 mm (.006 in), if ae=100 mm. However, if ae=80 mm (3.15 in),
for 0.15 mm (.006 in) chip thickness the feed should be already 0.24 mm/tooth
(.0094 ipt) that is approximately 10% more than the average value from the feed range.

Alternatively, instead of the feed values as a function of chip thickness, cutting tool producers often provide the data about the recommended feeds organized as a chart, which specifies a bounded area of application graphically (**Fig. 14**). The chart shows a range of possible feed from minimum (fz_{min}) to a limiting value depending on a cutting data parameter (usually depth of width of cut). If, as in **Fig. 14**, the parameter is the ratio of working depth of cut ap to maximum depth of cut ap_{max}, the starting feed in feed milling with ae=(0.6…0.8)×d may be simply found with the use of a thumb rule: the feed is the maximum feed that corresponds to the ratio in the chart.

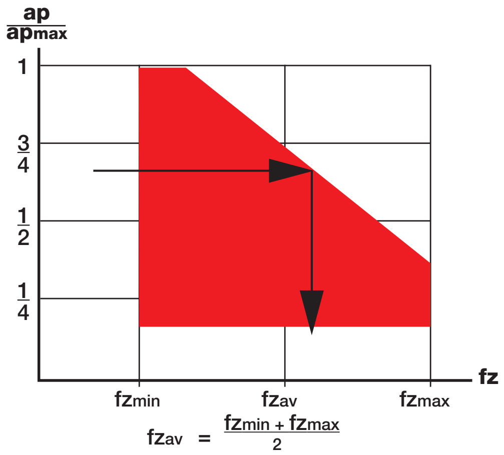


Fig. 14 A typical field of feeds in face milling for ae=(0.6…0.8)×d

Example

Find the feed for the example, which was discussed previously, if the range of usable feeds is specified by **Fig. 14** where fz_{min}=0.15 mm/tooth (.006 ipt) and fz_{max}=0.3 mm/tooth (.012 ipt).
ap_{max}=4.6 mm (.18 in) – the catalog data, ap/ap_{max}=3.5/4.6=0.76.
According to **Fig. 14**, arithmetic average feed fzav may be considered as the limiting value that corresponds to the above ratio.
Hence, following our rule of thumb, the starting feed, which was chosen as the arithmetic average, will be 0.225 mm/tooth (.009 ipt).
In the shown example, applying the rule of thumb method gave a result that can be accepted with some reserve. The received value is about 10% more than the starting feed, which was found with the use of chip thickness calculations previously.

In finding starting feeds, empirical rules led to acceptable results in relatively limited cases. However, calculating the feeds using required chip thickness is more accurate and therefore preferable, particularly as this calculation is simple.

Starting feed fz can be determined by several ways. For rough estimates, setting fz as the average value in the feed range from a catalog (a technical brochure), using rules of thumb such as in the outlined above example or own experience may be quite enough. A more accurate and less complicated method requires taking into consideration the chip thickness factor.

☛ *In any case, maximum chip thickness hmax should not exceed maximum feed fzmax specified for the considered inserts, solid carbide endmills or **MULTI-MASTER** heads*

Peripheral Milling

Table 15 gives averaged values of basic feed per tooth fo and corresponding parameters of chip thickness (hm, hmax) depending on the ratio between ae and d and a carbide grade for peripheral milling by indexable cutters and Table 16 – by solid carbide endmills and **MULTI-MASTER** heads.

Starting feed per tooth fz is defined by the following equations:
for 90° indexable cutters

$$fz=fzo\times Ks \qquad (5)$$

- for 90° SCEM and **MULTI-MASTER** heads

$$fz=fzo\times Ks\times Kf \qquad (6)$$

Where: fzo – basic feed per tooth
Ks – stability factor
Kf – tooth-strength factor (Table 17)

The stability factor here is set as below:

- Ks=1 for normal stability
- Ks=0.8-0.85 for unstable operations (high overhang, poor tool- or workholding, milling thin walls, etc.), and milling in unfavorable conditions.

Example

A manufacturer considers milling square shoulder by a **MULTI-MASTER** tool that carries a 20 mm (.750 in) Dia. cutting head, made from carbide grade IC308. The shoulder is 9.5 mm (.375 in) in depth and 5 mm (.2 in) in width. Operational stability is sufficient. Which feed per tooth (chip load) fz should be set as starting?

ae/d=5/20=0.25. fzo=0.085 mm/tooth
(Table 16), Kf=0.94 (Table 17). Ks is assumed as 1.
fz=0.085×0.94×1=0.08 (mm/tooth) or .031 ipt.

The average chip thickness in this case will be 0.042×0.94=0.04 (mm) (or .15 in), and the maximum chip thickness – 0.073×0.94=0.068 (mm) (or .026 in).

Table 15 Basic Feed per Tooth fzo and Chip Thickness Data for Peripheral Milling by Indexable Cutters

Ae/d (%)	AE	Parameters	Fzo, mm/tooth (ipt), and hm and hmax, mm (in), for carbide grade of inserts*						
			Main Grades			Complementary Grades			
			IC808 IC908	IC840	IC882	IC5820	IC380	IC830 IC928	IC330 IC328
0.05 (5%)	25.8°	hm	0.06 (.0023)	0.06 (.0023)	0.07 (.0027)	0.06 (.0023)	0.05 (.002)	0.07 (.0027)	0.07 (.0027)
		hmax	0.11 (.004)	0.11 (.004)	0.13 (.005)	0.11 (.004)	0.1 (.0039)	0.13 (.005)	0.13 (.005)
		Fzo	0.26 (.01)	0.26 (.01)	0.3 (.012)	0.26 (.01)	0.22 (.0087)	0.3 (.012)	0.3 (.012)
0.1 (10%)	**36°**	**Hm**	0.06 (.0023)	0.06 (.0023)	0.07 (.0027)	0.06 (.0023)	0.05 (.002)	0.07 (.0027)	0.07 (.0027)
		Hmax	0.11 (.004)	0.11 (.004)	0.13 (.005)	0.11 (.004)	0.1 (.0039)	0.13 (.005)	0.13 (.005)
		Fzo	0.19 (.0075)	0.19 (.0075)	0.22 (.0087)	0.19 (.0075)	0.16 (.0063)	0.22 (.0087)	0.22 (.0087)
0.15 (15%)	**45.6°**	**Hm**	0.06 (.0023)	0.06 (.0023)	0.07 (.0027)	0.06 (.0023)	0.05 (.002)	0.07 (.0027)	0.07 (.0027)
		Hmax	0.11 (.004)	0.11 (.004)	0.13 (.005)	0.11 (.004)	0.09 (.0036)	0.13 (.005)	0.13 (.005)
		Fzo	0.16 (.0063)	0.16 (.0063)	0.18 (.007)	0.16 (.0063)	0.13 (.0051)	0.18 (.007)	0.18 (.007)
0.2 (20%)	**53°**	**Hm**	0.067 (.0026)	0.067 (.0026)	0.076 (.003)	0.067 (.0026)	0.056 (.0022)	0.076 (.003)	0.076 (.003)
		Hmax	0.12 (.0047)	0.12 (.0047)	0.13 (.005)	0.12 (.0047)	0.09 (.0036)	0.13 (.005)	0.13 (.005)
		Fzo	0.15 (.0063)	0.15 (.0063)	0.17 (.0067)	0.15 (.0063)	0.125 (.005)	0.17 (.0067)	0.17 (.0067)
0.25 (25%)	**60°**	**Hm**	0.07 (.0027)	0.07 (.0027)	0.08 (.0031)	0.07 (.0027)	0.06 (.0023)	0.08 (.0031)	0.08 (.0031)
		Hmax	0.12 (.0047)	0.12 (.0047)	0.14 (.0055)	0.12 (.0047)	0.1 (.004)	0.14 (.0055)	0.14 (.0055)
		Fzo	0.14 (.0055)	0.14 (.0055)	0.16 (.0063)	0.14 (.0055)	0.12 (.0047)	0.16 (.0063)	0.16 (.0063)
0.3 (30%)	**66.4°**	**Hm**	0.07 (.0027)	0.07 (.0027)	0.08 (.0031)	0.07 (.0027)	0.06 (.0023)	0.08 (.0031)	0.08 (.0031)
		Hmax	0.12 (.0047)	0.12 (.0047)	0.127 (.005)	0.12 (.0047)	0.1 (.004)	0.127 (.005)	0.127 (.005)
		Fzo	0.127 (.005)	0.127 (.005)	0.14 (.0055)	0.127 (.005)	0.11 (.004)	0.14 (.0055)	0.14 (.0055)
0.4 (40%)	**78.5°**	**Hm**	0.07 (.0027)	0.07 (.0027)	0.08 (.0031)	0.07 (.0027)	0.06 (.0023)	0.08 (.0031)	0.08 (.0031)
		Hmax	0.108 (.0042)	0.108 (.0042)	0.122 (.0048)	0.108 (.0042)	0.098 (.0038)	0.122 (.0048)	0.122 (.0048)
		Fzo	0.11 (.0043)	0.11 (.0043)	0.125 (.0049)	0.11 (.0043)	0.1 (.004)	0.125 (.0049)	0.125 (.0049)
0.5 (50%)	**90°**	**Hm**	0.075 (.003)	0.075 (.003)	0.088 (.0035)	0.075 (.003)	0.065 (.0025)	0.088 (.0035)	0.088 (.0035)
		Hmax	0.11 (.0043)	0.11 (.0043)	0.125 (.0049)	0.11 (.0043)	0.09 (.0035)	0.125 (.0049)	0.125 (.0049)
		Fzo	0.11 (.0043)	0.11 (.0043)	0.125 (.0049)	0.11 (.0043)	0.09 (.0035)	0.125 (.0049)	0.125 (.0049)
0.75 (75%)	**120°**	**Hm**	0.095 (.0037)	0.095 (.0037)	0.11 (.0043)	0.095 (.0037)	0.078 (.0031)	0.11 (.0043)	0.11 (.0043)
		Hmax	0.11 (.0043)	0.11 (.0043)	0.125 (.0049)	0.11 (.0043)	0.09 (.0035)	0.125 (.0049)	0.125 (.0049)
		Fzo	0.11 (.0043)	0.11 (.0043)	0.125 (.0049)	0.11 (.0043)	0.09 (.0035)	0.125 (.0049)	0.125 (.0049)
1 (100%)	**180°**	**Hm**	0.1 (.004)	0.1 (.004)	0.11 (.0043)	0.1 (.004)	0.09 (.0035)	0.11 (.0043)	0.11 (.0043)
		Hmax	0.1 (.004)	0.1 (.004)	0.11 (.0043)	0.1 (.004)	0.09 (.0035)	0.11 (.0043)	0.11 (.0043)
		Fzo	0.1 (.004)	0.1 (.004)	0.11 (.0043)	0.1 (.004)	0.09 (.0035)	0.11 (.0043)	0.11 (.0043)

* The table values are given in metric units with their US customary equivalents in brackets: fz in mm/tooth (ipt), hm and hmax in mm (in)

Avoid misunderstanding: titanium beta (β)
An incorrect interpretation of shop talk may cause serious misunderstandings and even wrong decisions. In many cases, "titanium-beta", so freely used in manufacturing, does not mean β-alloys but β-(beta-) annealed α-β-alloys, in particular for the "universal" Ti-6Al-4V.

Table 16 Basic Feed per Tooth fzo and Chip Thickness Data for peripheral Milling by SCEM and MULTI-MASTER Heads

Ae/d (%)	AE	Parameters	Fzo, mm/tooth (ipt), and hm and hmax, mm (in), For carbide grade of SCEM and heads*			
			Main grades		Complementary grades	
			IC900 IC908	IC300 IC308	IC902**	IC903**
0.03 (3%)	19.9°	hm	0.04 (.0016)	0.042 (.0016)	0.035 (.0013)	0.037 (.0014)
		hmax	0.078 (.0031)	0.083 (.0032)	0.068 (.0027)	0.073 (.0029)
		fzo	0.23 (.009)	0.245 (.0096)	0.2 (.0079)	0.215 (.0085)
0.05 (5%)	25.8°	hm	0.04 (.0016)	0.041 (.0016)	0.035 (.0013)	0.037 (.0014)
		hmax	0.078 (.0031)	0.08 (.0031)	0.067 (.0026)	0.072 (.0028)
		fzo	0.18 (.0071)	0.185 (.0073)	0.155 (.0061)	0.165 (.0065)
0.1 (10%)	36°	hm	0.041 (.0016)	0.042 (.0016)	0.036 (.0014)	0.038 (.0015)
		hmax	0.076 (.003)	0.078 (.0031)	0.067 (.0026)	0.07 (.0027)
		fzo	0.13 (.0051)	0.133 (.0052)	0.115 (.0045)	0.12 (.0047)
0.15 (15%)	45.6°	hm	0.04 (.0016)	0.043 (.0017)	0.035 (.0013)	0.037 (.0014)
		hmax	0.075 (.0029)	0.078 (.0031)	0.065 (.0026)	0.068 (.0027)
		fzo	0.105 (.004)	0.11 (.0043)	0.091 (.0036)	0.095 (.0037)
0.2 (20%)	53°	hm	0.04 (.0016)	0.041 (.0016)	0.036 (.0014)	0.038 (.0015)
		hmax	0.07 (.0027)	0.074 (.0029)	0.064 (.0025)	0.067 (.0026)
		fzo	0.09 (.0035)	0.093 (.0037)	0.08 (.0031)	0.084 (.0033)
0.25 (25%)	60°	hm	0.04 (.0016)	0.042 (.0016)	0.036 (.0014)	0.038 (.0015)
		hmax	0.07 (.0027)	0.073 (.0028)	0.063 (.0025)	0.066 (.0026)
		fzo	0.081 (.0032)	0.085 (.0033)	0.073 (.0029)	0.076 (.003)
0.3 (30%)	66.4°	hm	0.04 (.0016)	0.043 (.0016)	0.036 (.0014)	0.038 (.0015)
		hmax	0.068 (.0027)	0.071 (.0028)	0.06 (.0024)	0.063 (.0025)
		fzo	0.073 (.0029)	0.078 (.0031)	0.065 (.0026)	0.069 (.0027)
0.4 (40%)	78.5°	hm	0.04 (.0016)	0.043 (.0016)		
		hmax	0.063 (.0025)	0.067 (.0026)		
		fzo	0.064 (.0043)	0.068 (.0027)		
0.5 (50%)	90°	hm	0.04 (.0016)	0.042 (.0016)		
		hmax	0.057 (.0022)	0.06 (.0024)		
		fzo	0.057 (.0022)	0.06 (.0024)		
0.75 (75%)	120°	hm	0.049 (.0019)	0.052 (.002)		
		hmax	0.057 (.0022)	0.06 (.0024)		
		fzo	0.057 (.0022)	0.06 (.0024)		
1 (100%)	180°	hm	0.057 (.0022)	0.06 (.0024)		
		hmax	0.057 (.0022)	0.06 (.0024)		
		fzo	0.057 (.0022)	0.06 (.0024)		

* The table values are given in metric units with their US customary equivalents in brackets: fz in mm/tooth (ipt), hm and hmax in mm (in)

** The grades are recommended mainly for HSM by trochoidal milling and finish operations

Table 17 Tooth-Strength Factor Kf for SCEM and MULTI-MASTER Heads

Nom. diameter D	mm	24<d≤32	19<d≤24	14<d≤19	9<d≤14	d≤9
	(in)	(.945<d≤1.250)	(.750<d≤.945)	(.551<d≤.750)	(.354<d≤.551)	(d≤.354)
Kf		1	0.94	0.88	0.78	0.65

Example

A producer of jet engines plans to apply ISCAR's 12 mm dia. 7 flute solid carbide endmills for machining slots in titanium integrally bladed rotors (blisks) by trochoidal milling technique. The titanium grade is Ti-6Al-4V, HRC 33-35. In milling slots, the width of cut is 1-1.2 mm, and the depth of cut is variable. The selected endmills are made from cemented carbide IC902. The machine tools intended for this operation are in good condition and clamping rotor rigidity is high. Estimate cutting data for calculating cycle time.
According to equation (1) cutting speed Vc=Vo×Km×Ke×Ks.
Vo=75 m/min (246 sfm) from Table 8,
Km=1 (Table 9),
Ke=1.5 (Table 10),
Ks=1 (operational stability is sufficient),
Vc=75 ×1×1.5×1=112 (m/min) (370 sfm),
n=2971 rpm.
fz=fzo×Ks×Kf (equation (6)),
fzo=0.115 mm/tooth (0.0045 ipt) from
Table 16,
Kf=0.78 (Table 17),
fz=0.115×1×0.78=0.09 (mm/tooth) (.0035 ipt).

Feed speed Vf=0.015×7×2971=312 (mm/min) (12.3 ipm).

Example

Find cutting data for edging operation to be performed by a 63 mm (2.5 in) dia. extended flute endmills with DIN69871 tapered shanks carrying T490 LNMT/LNHT 13... tangentially clamped inserts. The endmill has 4 flutes (effective teeth). The workpiece material is titanium alloy "triple 5" VST 5553 (Ti-5Al-5Mo-5V-3Cr). The width of cut varies from 6 to 8 mm (.236-.315 in), the depth of cut is around 50 mm (2 in). The inserts are made from carbide grade IC882. Cutting conditions are estimated as stable. Ratio ae/d is accepted as 0.12 in average.
Usually, edging relates to removing a skinned material. As a rule, the extended flute tools with integral shank feature high overhang. Based on these conditions, the type of machining is assumed as hard (Table 6).
Vo=38 m/min (125 sfm) (Table 7), Km=0.51 (Table 9), Ke=1.3 (Table 10), Ks=1.
Vc=38 ×0.51×1.3×1=25 (m/min) (83 sfm) (equation (1)),
N=126 rpm.
fzo≈0.2 mm/tooth (.0079 ipt) (Table 15, a value that lays between data corresponding to ae/d=0.15 and ae/d=0.1).
fz=0.2×1=0.2 mm/tooth (.0079 ipt).
Vf=0.2×4×126=100.8 (mm/min) (3.97 ipm).
The average chip thickness and the maximum chip thickness are 0.07 mm (.0027 in) and 0.13 mm (.005 in) correspondingly (Table 15).

Face Milling

In face milling, starting feed per tooth fz is calculated by equation (7):

$$fz = fzo \times 1/\sin\chi \times Ks \qquad (7)$$

Here: fzo – basic feed per tooth (Table 18),
χ – cutting edge (entering) angle,
Ks – stability factor.

Face milling features considerable width of cut: normally ae=(0.6…0.8)×d. Therefore, for quick estimation fzo may be set as the basic feed per tooth from Tables 15 and 16 that corresponds ae/d=0.75. The appropriate values of fzo for indexable milling cutters are shown in Table 18.

Table 19 gives already calculated values of 1/sinχ for typical cutting edge angles of face mills.

Table 18 Basic Feed per Tooth fzo and Chip Thickness Data for Indexable Face Milling

	Fzo, mm/tooth (ipt), and hm and hmax, mm (in), for carbide grade of inserts*						
	Main grades			**Complementary grades**			
Parameters	**IC808 IC908**	**IC840**	**IC882**	**IC5820**	**IC380**	**IC830 IC928**	**IC330 IC328**
Hm	0.095 (.0037)	0.095 (.0037)	0.11 (.0043)	0.095 (.0037)	0.078 (.0031)	0.11 0043)	0.11 (.0043)
Hmax	0.11 (.0043)	0.11 (.0043)	0.125 (.0049)	0.11 (.0043)	0.09 (.0035)	0.125 (.0049)	0.125 (.0049)
Fzo	0.11 (.0043)	0.11 (.0043)	0.125 (.0049)	0.11 (.0043)	0.09 (.0035)	0.125 (.0049)	0.125 (.0049)

* The table values are given in metric units with their US customary equivalents in brackets:
fz in mm/tooth (ipt), hm and hmax in mm (in)

Table 19 Calculated Values of (1/Sinχ) for Typical Cutting Edge Angles

X	**90°**	**75°**	**65°**	**60°**	**45°**	**30°**
1/sinχ	1	1.03	1.1	1.15	1.4	2

Cutting edge angle and lead angle are not synonyms!
The "cutting edge angle" is the angle between the main cutting edge of a milling cutter and the plane containing the direction of feed motion. "Lead angle" (or "approach angle") is the angle complementary to the cutting edge angle, i.e. the sum of both these angles is 90°. For a typical face milling cutter, the cutting angle is the angle between the cutting edge and the plane, which the cutter generates. If this angle is 60°, then the lead angle will be 30°. The cutting edge angle and the lead angle are equal only for 45° milling cutters. The term "lead angle" is more commonly employed in the U.S., while "approach angle" is often used in Europe and Japan.

Solid carbide endmills and the majority of **MULTI-MASTER** tools are not intended for face milling. However, there are **MULTI-MASTER** replaceable heads (for example, MM FM, **Fig. 15**) and special solid carbide cutters that were designed especially for face milling applications.

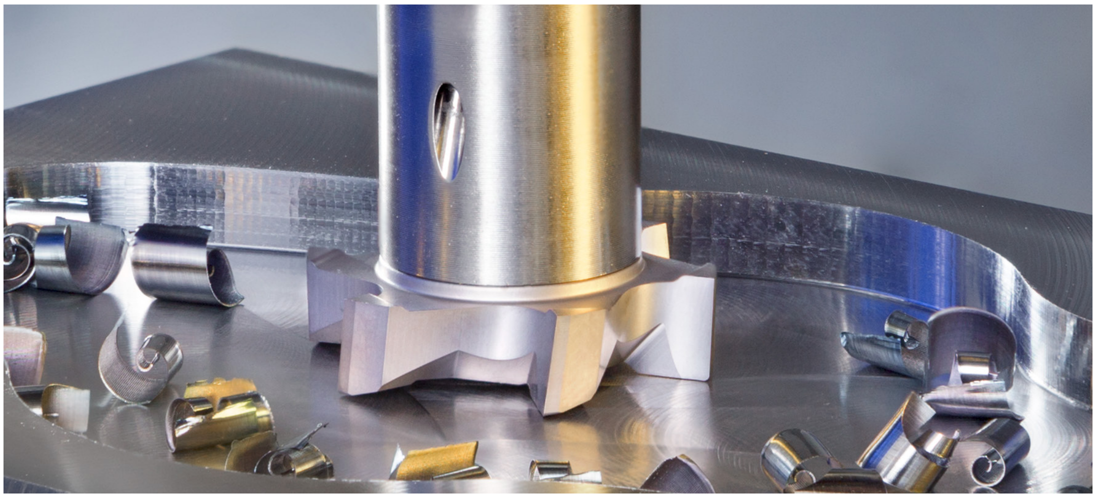


Fig. 15 MULTI-MASTER face milling head MM FM

In this case starting feed per tooth fz can be found with the use of equation (7a):

$$fz=fzo\times Kf\times 1/\sin\chi \times Ks \qquad (7a)$$

Here: ***fzo*** – basic feed per tooth (Table 20),
Kf – tooth-strength factor (Table 17)
χ – cutting edge (entering) angle.

Example

A producer of titanium components needs to machine a plane surface of a large-size part, which is made from Ti-6Al-4V. 200 mm dia. milling cutter SOF45 8/16-D200-14-60R carrying inserts ONHU 050500-PL IC830 is intended for applying in finish milling the surface. The depth of cut and the width of cut are 1.5 mm and 160 mm correspondingly. Which cutting speed and feed should be set?

Cutting edge angle χ=45°, ae/d=0.8, the type of machining is considered as light (Table 6), and operational stability is assumed as sufficient.

For carbide grade IC830 Vo=55 m/min or 180 sfm (Table 7), Km=1 (Table 9),Ke≈0.92 (Table10), Ks=1.
Hence, from equation (1) Vc=55×1×0.92×1=50.6 (mm/min) ≈51 m/min (167 sfm).
n=1000×51/π/200=81 (rpm).

Table 18 gives fzo=0.125 mm/tooth (.0049 ipt), 1/sin45°=1.4 (Table 19); and using equation (7)
fz=0.125×1.4×1=0.175 (mm/tooth) (.0069 ipt).
Vf=0.175×14×81=198 (mm/min) (7.82 ipm).

Table 20 Basic Feed per Tooth fzo and Chip Thickness Data for Face Milling by Solid Carbide and MULTI-MASTER Tools

Parameters	Fzo, hm and hmax for carbide grade	
	Main grades	
	IC900 IC908	IC300 IC308
hm	0.049 (.0019)	0.052 (.002)
hmax	0.057 (.0022)	0.06 (.0024)
fzo	0.057 (.0022)	0.06 (.0024)

* The table values are given in metric units with their US customary equivalents in brackets: fz in mm/tooth (ipt), hm and hmax in mm (in)

Tools for Milling Titanium Efficiently

ISCAR provides a rich variety of tools that are specifically designed for milling titanium. This section is intended for quickly selecting a suitable tool family for the most typical milling operations that require machining titanium components.

Indexable Extended Flute Cutters for Milling Deep Shoulders and Wide Edges

A considerable part of milling titanium, especially in producing large-size parts, relates to removing a high volume of material during rough machining of various cavities and pockets. A common way to do this is by applying indexable extended flute (long-edge) cutters. These mainly have a cutting edge angle of 90° and perform peripheral milling of deep shoulders. In addition, these cutters are the first-choice tools for machining wide edges of workpieces (edging).

Referring to the most popular ISCAR families of extended flute cutters, Table 21 and Table 22 prioritize the families and Table 23 specifies their general design features.

Table 21 Indexable Tool Families for Milling Deep Shoulders

		1st choice	2nd choice	Possible
MILLING DEEP SHOULDERS	Rough	**XQUAD EXTENDED FLUTE**	HELITANG T490 LINE	**MILLSHRED** P290 LINE
	Semifinish	**HELITANG FIN LNK**	XQUAD EXTENDED FLUTE	**MILLSHRED** P290 LINE

Table 22 Indexable Tool Families for Milling Wide Edges (Edging)

		1st choice	2nd choice	Possible
MILLING WIDE EDGES (EDGING)	Rough	**HELITANG T490 LINE**	XQUAD EXTENDED FLUTE	**MILLSHRED** P290 LINE
	Semifinish	**HELITANG FIN LNK**	XQUAD EXTENDED FLUTE	**MILLSHRED** P290 LINE

Chip splitting and chip chopping in milling titanium

The desire to improve performance and increase the dynamic behavior of extended flute cutters led to the use of inserts with a chip splitting action. The cutting edges of these advanced inserts feature specially designed chip splitting grooves. Further developments have resulted in the introduction of inserts with chip crushing ability, that literally chop chips into small segments. Advances in powder metallurgy allow the production of tough, sintered chip crushing inserts with high-strength shredded cutting edges, such as ISCAR's indexable ***MILLSHRED*** *P290 - a family of extended flute cutters that adheres to the more common concept of clamping inserts radially. The main distinctive feature of this family is the serrated cutting edge of the inserts, which chops or shreds the chips. Unlike many chip splitting inserts that are currently available, the P290 one-sided inserts, which possess two indexable serrated cutting edges, do not require special instructions for mounting in the* ***MILLSHRED*** *extended flute cutters, and are simply secured in any pocket. This operator-friendly feature simplifies tool assembly and eliminates errors in insert indexing, which may cause tool destruction.*

Table 23 Indexable Extended Flute Cutter Families: General Data*

Family	Diameter range		Design configuration				Insert Clamping	Insert			
			Endmills with shank**		Shell mills	Flex. tooling And heads		Type (O/D***)	No. of Cut. edges	Cut, edge form	
	mm	in	Cylindrical	Tapered						Smooth	Chipslitted
XQUAD	50-100	2.0-4.0			●		radial	O	4	●	
HELITANG T490 LINE	32-80	1.0-4.0	●	●	●	●	tangential	D	4	●	●
HELITANG FIN LNK	50-80	1.97-3.15		●			tangential	O	2	●	
MILLSHRED P290 LINE	25-100	1.0-4.0	●	●	●	●	radial	O	2	●	●

* at the time of writing
** The type of the shank is indicated in the columns
*** O-one sided, D-double-sided

Radial or tangential?
Generally, the concept of a tool carrying one-sided radially clamped inserts is more common in the design of the indexable extended flute cutters for milling titanium. A one-sided radially clamped (laydown) insert provides more options for varying cutting geometry and ensuring necessary relief angles. Cutters carrying these inserts usually feature greater chip gullet than tools with tangential inserts and, therefore, better chip evacuation when milling with high metal removal rate, especially in machining deep cavities and pockets. Also, they usually have higher ramping abilities. This explains why the radial concept prevails in extended flute cutters for milling titanium. ISCAR's family of ***HELITANG T490*** *extended flute cutters, primarily tools with size 13 inserts, is an exception to this general rule. The principle of tangential clamping contributes to improving the cutter rigidity and optimal loading of a screw to secure the inserts while usually ensuring a higher insert density that results in increased feed rate.* ***HELITANG T490*** *extended flute cutters show their worth in edging – rough milling the wide edges of titanium workpieces, where the depth of cut is considerable, but the width of cut is not too large. Under these conditions there is no problem with chip evacuation, and the extended flute cutter carrying tangentially clamped inserts wholly meets the requirements of high-efficiency roughing.*

☛ *Chip splitting geometry may be a good solution for rough milling in unstable conditions (high overhang, poor workholding etc.). In many such cases, chip splitting cutters demonstrates better dynamic behavior that results in better performance.*

Fig. 16 A “fraze”

“Raking by fraze”
Chip chopping is the main principle of the ISCAR ***MILLSHRED*** *P290 family. The cutting action of a shredded P290 mill can be compared with a fraze or a rotary rake (**Fig. 16**). If you need to loosen soil, the fraze will require less effort than a hoe. In our case, if you overlap the teeth of your “fraze”, this will reduce the production of cusps and will improve surface quality. The hand movements in **Fig. 16** illustrate this action.*

Milling With High Pressure Coolant (HPC)

There are no strict definitions of high and ultra high pressure coolant (HPC and UHPC correspondingly). Traditionally, machine tools feature coolant supply at 10-15 bar (145-217 psi) pressure. This level is now considered as low pressure. Some CNC machine tool equipment producers manufacture what are known as “medium pressure” pumps; these have values of up to 50 bar (725 psi). Various modern machining centers have the option to supply coolant at rates of 70-80 bar (1000-1200 psi), which is considered as high pressure coolant. High pressure coolant features coolant flow pinpointed to its target. Today the industry adopts machining with ultra high pressure coolant that relates to pressure values of 100-200 bar (1450-2900 psi) and even higher.

There is growing interest in cutting with HPC due to the ultimate performance of this method, especially when heat generation rate during machining is high. As milling titanium relates to processes with intensive heat emission, it is no wonder that HPC has focused titanium manufacturers’ attention.

Heat generation is a permanent feature of machining. If heat generation is intensive, the conventional low pressure coolant forms a vapor layer on the surfaces of a tool and a workpiece. This layer acts as heat sealing, producing an insulating barrier and making heat transfer harder, which significantly shortens tool life.

Pinpointed high pressure coolant penetrates the barrier and helps to overcome the problem.

- HPC improves tool life of a cutting edge due to reducing oxidation and adhesion wear and increasing crack strength.
- HPC chills chips quickly, making them hard and brittle. The chips become thinner and smaller, and they break away from the workpiece more easily. High-speed coolant flow removes the chips. This significantly improves chip evacuation and prevents chip re-cutting.
- HPC improves chip evacuation because the chips diminish in size, and the high-velocity coolant flow takes them away easily. It allows the design of cutters with smaller chip gullet, leading to a higher number of cutter teeth. Effective cooling reduces the temperature in the cutting zone, ensuring an increased angle of engagement AE (i.e. width of cut ae).

☛ *Overall, HPC provides a good solution for increasing cutting speed and feed rate for boosting productivity, which is based on:*
- *transforming improved tool life in higher cutting speed,*
- *increasing feed rate by using a tool with more teeth comparing with the tool of same diameter that is intended for low pressure coolant,*
- *milling with greater width of cut.*

In view of this, the up-to-date machining centers, which MTB companies offer to producers of titanium components, are suitable for HPC supply. As practice shows, machining with HPC can even double cutting speed in comparison with classical As shown in practice, Accurate value for increasing cutting speed depends on specific factors. To estimate initial cutting speed in milling with HPC V_{HP}, it is recommended to use the following equation (8):

$$V_{HP}=(1.2\ldots1.3)\times V_c, \qquad (8)$$

where Vc – Initial cutting speed, given by equation (1), or determined with the use of (Table 6).
By assuming that the increase in V_{HP} is 25% in average, this equation
after substituting the terms of equation (1) will look as follows:

$$V_{HP}=1.25\times V_o\times K_m\times K_e\times K_s \quad (8a)$$

As in case of finding starting speed in milling in general, basic cutting speed Vo, factors: Km (machinability), Ke (engagement) and Ks (stability) are (also) found in Tables 7-10. Stability factor Ks is assumed as 1 for normal stability, and 0.7…0.8 for milling in unstable conditions.

Example

Find initial cutting data for rough milling deep square shoulder in a workpiece from titanium Ti-6Al-4V ELI. The applied tool is ISCAR's 80 mm (3 in) dia. indexable extended cutter carrying inserts made from carbide grade IC5820. The cutting parameters are as below:
depth of cut ap=85 mm (3.35 in), width of cut ae=20 mm (.79 in).
The machine intended for the milling operation is a vertical machining center with HPC option, and the cutter has inner channels for high pressure coolant supply. Operational stability can be estimated as high.

Vo=43 m/min (141 sfm). (Table 7, hard type of machining)
Km=1.1 (Table 9).
Ke=1.2 (Table 10).
Ks=1.
Vc=43×1.1×1.2×1=52 (m/min) or 170 sfm.
Hence, VHP=(1.2…1.3) ×Vc ≈ 65 m/min=
213 sfm.
From Table 15 for ae/d=0.25 fzo=0.14 mm/tooth (0.0055 ipt) and
fz=fzo×Ks=0.14×1=0.14 (mm/tooth)
or 0.0055 ipt.

There are different factors that influence HPC supply through the body of a milling cutter, which tool engineers take into account when designing the cutter. In indexable milling, from the application point of view, attention should be drawn to the coolant outlets. They are formed by nozzles, screwed in appropriate holes that open into chip gullets (**Fig. 17, 18)**.

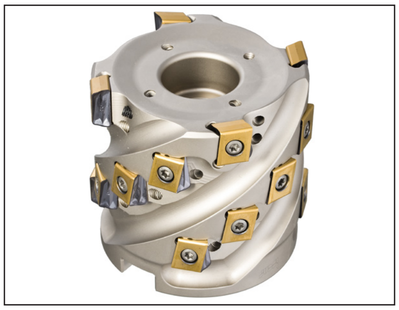


Fig. 17 A T490 extended flute cutter Intended for HPC. The cutter design Enables nozzles to be mounted in the Coolant supply hole outlets

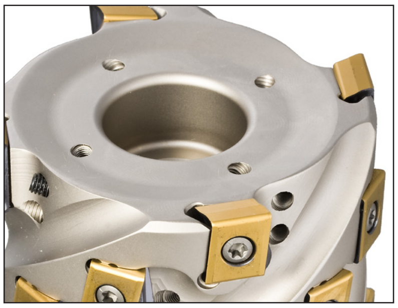


Fig. 18 A nozzle mounted in a coolant supply hole near a T490 extended flute cutter face-cutting insert

Why nozzles?

There are several reasons for using nozzles as outlets in indexable extended flute cutters. Firstly, a technological aspect. The required outlet diameter usually lays within 0.6…2 mm range. Drilling these "pinholes" deep enough in a relatively hard steel body is quite difficult. It makes sense to increase the diameter, cut a thread near the opening, and screw in an appropriate nozzle. Secondly, a control reason. Varying outlet diameters by changing the nozzles allows optimizing coolant flow that depends on the characteristics of an HPC pump and the number of the nozzles in a cutter. Also, if the cutter does not mill to its maximum depth of cut, the coolant holes near the non-working inserts can be easily plugged.

The nozzle diameter is less than the diameter of the hole. In accordance with the laws of flow mechanics, input and output relations for a fluid velocity exist. The difference in diameters increases the coolant velocity, and the coolant leaves the nozzle with higher velocity than enters in it. On the other hand, there is a certain relationship between pressure, velocity and flow rate for fluid, e.g. for coolant in our case. Increasing the nozzle diameter requires higher flow rate to ensure a necessary pressure level. A growth of the number of the nozzles has the same effect.

For efficient application of indexable extended flute milling cutters with HPC supply through the cutter body, the following two points should be considered:

1. If a turning tool has one cutting edge and one nozzle is sufficient for coolant supply, the extended flute cutter features several cutting blades. Each blade is produced by a set of replaceable inserts, located one after another. A pinpointed jet of coolant to the blade areas demands at least one nozzle per every insert. It means that the characteristics of an HPC pump should meet the required flow rate that considerably differs from turning cases.
2. If depth of cut is smaller than the length of the cutting blade, there is no need to supply coolant to the inserts that are not involved in cutting. It is recommended to unscrew the nozzles corresponding to these inserts from the holes, and then close the holes by plugs or standard set screws. This is a simple and effective way for improving performance.

Standard or special (tailor-made)?

Indexable HPC extended flute cutters are intended for removing a high volume of metal. They work in hard cutting conditions under heavy load that is usually combined with a substantial tool overhang. This raises the question as to which is preferable, assembly from a standard cutter and standard modular toolholding units such as arbors, holders, reducers etc., or a special (tailor-made) integral tool with direct adaptation to the spindle? Although assembly from standard elements looks reasonable, titanium component manufacturers prefer the integral design in many cases. This configuration provides maximum rigidity and ensures more productive milling. Specific tool diameters, cutting lengths, and overhang, as well as adaptations that vary from one manufacturer to another, increase demands for various tailor-made HPC milling cutters.

Fast Feed Milling Titanium

Fast feed milling (FF), also referred to as high feed milling (HFM), is usually associated with productive rough machining steel and cast iron. However, this effective method of rough milling may be successfully applied to manufacturing titanium components.

Rather than using a traditional high metal removal technique – milling with considerable depths and widths of cut – FF proposes machining with similar widths of cut but with a much smaller depth of cut. FF cutters feature small cutting edge angles that allow significant increasing in feed per tooth fz and therefore feed speed Vf due to the effect of chip thinning (**Fig. 19**).

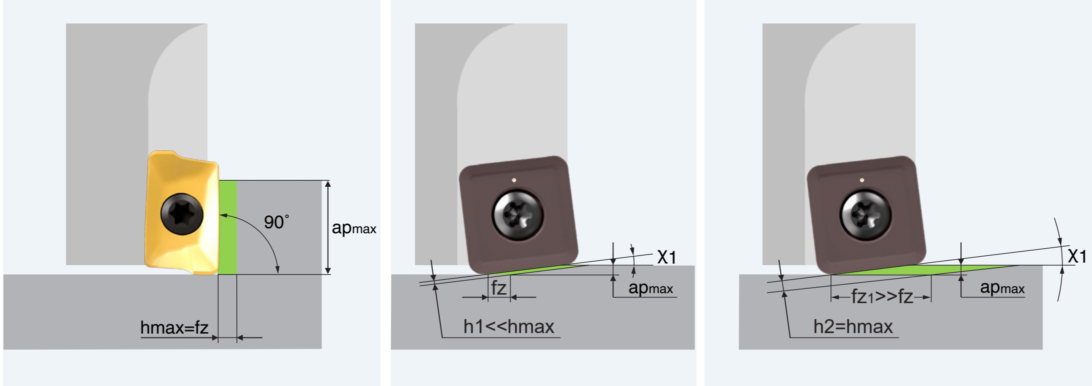


Fig. 19 Geometrical relations between feed per tooth, cutting edge angle and chip thickness

Another distinctive feature of fast feed milling is reducing the bending moment, which an FF tool takes. The small cutting angle leads to minimizing the radial effect of the cutting force and maximizing its axial influence. The bending moment depends on the force acting on the tool perpendicular to its axis. This bending force is the resultant of the radial and the tangential components of the cutting force, substantially decreasing the bending moment. The axial force acts towards the spindle axis, i.e. the direction of maximum machine tool rigidity. The result – improved milling stability, reduced vibration and increased productivity. Decreasing the bending force is especially important in machining titanium, due to the already mentioned “springiness” of the material.
The classical cutting edge of the FF tool is an arc or large diameter. Today, this geometry mainly features solid carbide endmills and replaceable heads. The cutting edge of the indexable tools is usually one or two chords of the arc.

ISCAR has a rich variety of FF tool families: indexable mills, SCEM and **MULTI-MASTER** cutters. Indexable tools carry inserts that differ in type (one- and double sided), shape (trigon, quadrihedral, hexagonal etc.) and size.
Table 24 provides general guidelines for choosing the most suitable indexable family for fast feed milling titanium, from ISCAR’s standard product line.

A new birth of FF tools

Originally, FF tools were indexable cutters of relatively large nominal diameters. Advances in multi-axis grinding machines enabled development of FF geometry in tools of solid design: SCEM and replaceable heads of considerably less diameters. Further attempts to find a cost-saving alternative to more accurate solid tools resulted in a small-size indexable solution.
For high feed milling titanium, ISCAR proposes the recently-introduced ***MICRO3FEED*** *– a family of multi-flute indexable FF cutters with a 10 mm minimum diameter.*

Table 24 Indexable Extended Flute Cutter Families: General Data*

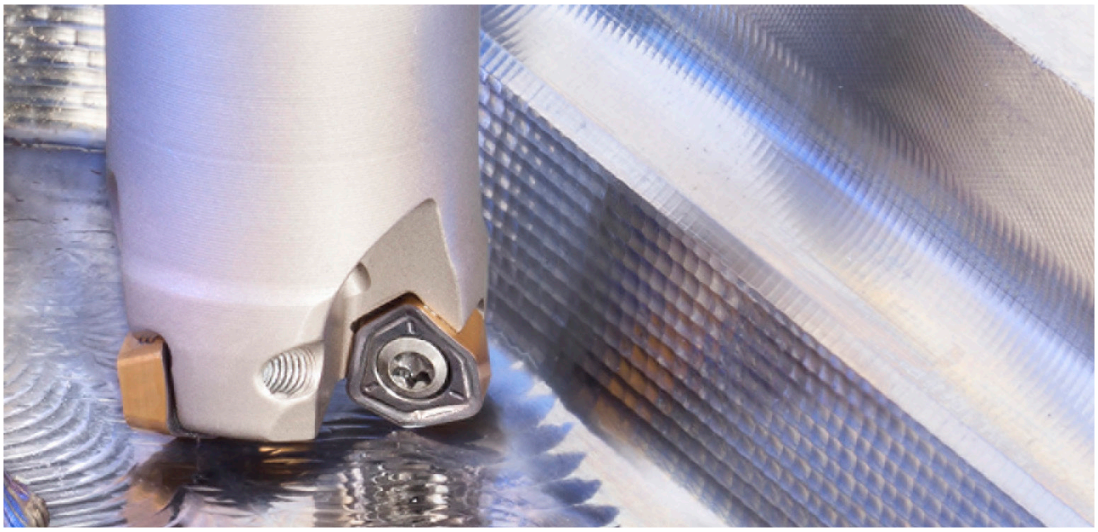
Diameter Range	mm	10-16	20-40	50-63	80-100
	in	.315-.625	.750-1.500	2.000-2.500	3.000-4.00
1st choice		**MICRO3FEED (T)**	**HELI6FEED (HP)**	**TANG4FEED (ER)**	**MILL4FEED (HP)**
2nd choice			MILL4FEED (HP)	MILL4FEED (HP)	TANG4FEED (ER)
Possible			LOGIQ4FEED (HP)	HELI6FEED (HP)	HELI6FEED (HP)

* The recommended type of the insert chipformer is shown in brackets.

ISCAR's FF solid carbide endmills and **MULTI-MASTER** heads are represented by several families. A general description of the families is given in Table 25.

Table 25 FF SCEM and MULTI-MASTER Heads

Family	Designation	Range of diameters mm	in	Number of teeth
SOLIDFEEDMILL	EFF-S	1-20	.250-.750	2, 4
MULTIFEEDMASTER	MM EFF	8-25	.472-1.000	4, 6
MULTIFEEDMASTER	MM FF	10-20	.375-.750	2



Applications

The main operations performed by FF tools are rough milling pockets and cavities ("pocketing"), pre-shaping complex surfaces ("profiling") and plane faces ("facing"). Some of the tools feature side plunge milling capabilities ("side plunging"). Milling pockets and cavities are the most common cutting operations performed in producing titanium parts.
It is worth noting however, that fast feed face milling titanium is still not as popular as steel. The reason is, again, heat. Thick chips, produced by an FF tool, makes heat removal from the cutting zone more difficult.
Also, face mills generally feature relatively large diameters that increase the contact of an insert with the machined material. This results in intensifying the heat load on the cutting edge and shortening tool life.
By contrast, applying FF tools to pocketing enables decreasing the bending force in high feed milling and therefore this method is recommended for rough machining pockets and cavities in thin-walled and low-rigidity parts.

Fast Feed Facing
Initially the FF cutters were considered mainly as tools for productive rough milling of cavities and punches in die and mold applications. The FF approach was later applied to face milling.
Fast feed face milling ("fast feed facing" or, simply, "triple F") with the use of indexable cutters, opened another application field for FF tools.

☛ *An FF tool generates a machined surface with cusps. The cusp height diminishes with reducing width of cut ae. It is recommended that the width of cut be no more than diameter DC (**Fig. 20**) to prevent tooth overloading, because of excess machining allowance in the produced cusps.*

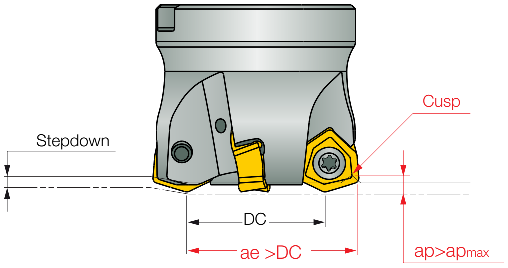


Fig. 20 Width of cut and cusps in fast feed milling

In CNC programming, an FF tool is often specified as a milling cutter with a corner radius. The radius is calleda "radius for programming" (R in **Fig. 21**). It defines the maximum thickness of a cusp – a mismatch, produced by such specification (correspondingly t in the same figure).

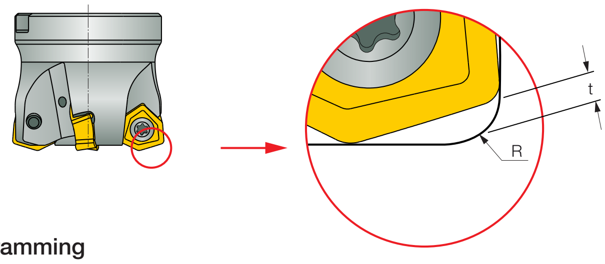


Fig. 21 Radius for programming

Carbide Grades

HFM demands tough carbide grades. In indexable milling, IC882 is the first choice and IC330, IC328, and IC840 are reasonable options. Solid carbide FF endmills are produced from grades IC903 and IC902, and FF **MULTI-MASTER** heads – from IC903 and IC908.

Initial Cutting Data

Initial cutting speed Vc can be found using equation (1), where the basic cutting speed and factors, which the equation uses, are in Tables 7-10. Basic cutting speed Vo should be chosen from data for medium (M) type of machining. As in the previous cases, stability factor Ks is defined as 1 for normal stability and 0.-0.8 for unstable machining conditions.

Note. In milling cavities and pockets, in many cases an FF tool starts from cutting solid material by ramping down (with AE=180°) and then continues to machine the shoulder. For a rough calculation of Vc it is enough to simplify equation (1) by setting engagement factor Ke as 1.

Starting feed per tooth fz for indexable FF tools is specified by equation (5):

$$fz=fzo\times Ks$$

Table 26 provides basic feeds depending on carbide grades of inserts, and Table 27 – for SCEM and **MULTI-MASTER** heads.

Table 26 Basic Feed fzo for Indexable FF Tools

Family	Tool Designation	Insert Size	Cutting Edge angle	Fzo, mm/tooth (ipt), For carbide grades of FF inserts		
				IC882, IC330, IC328	IC830, IC840, IC5820	IC808, IC908
MICRO3FEED	FFT3	03	17°	0.38 (.015)	0.35 (.0137)	0.33 (.013)
HELI6FEED	FF EWX/FWX	04	17°	0.4 (.016)	0.38 (.015)	0.36 (.0142)
		05		0.4 (.016)	0.38 (.015)	0.36 (.0142)
		07		0.45 (.0177)	0.42 (.0165)	0.4 (.016)
		08		0.45 (.0177)	0.42(.0165)	0.4 (.016)
MILL4FEED	FFQ4	09	12°	0.55 (.0217)	0.51 (.02)	0.49 (.0193)
		12	9°	0.7 (.0275)	0.65 (.0256)	0.62 (.0245)
		17	14°	0.47 (.0185)	0.44 (.0173)	0.42 (.0165)
LOGIQ4FEED	FFX4	04	17°	0.45 (.0177)	0.42 (.0165)	0.4 (.016)
TANG4FEED	FFV	07	16°	0.47 (.0185)	0.44 (.0173)	0.42 (.0165)

* The recommended type of the insert chipformer is shown in brackets.

Example

It is planned to use the HFM method for rough milling the cavity in a titanium part. The titanium grade is solution treated and aged Ti-6Al-4V. The cavity dimensions (length L × width W × depth H) are 220 × 110 × 70 mm (8.66 × 4.33 × 2.75 in). The technologist of a company machining division that has **LOGIQ4FEED** fast feed shell mill FFX4 FD040-6-16-04 on hand, decided to apply it and machine the cavity by ramping down including ramping by helix. The mill carries inserts FFX4 XNMU 040310HP IC882. Operational stability is considered as normal. Which cutting data should be set?

Basic cutting speed Vo=43 m/min (141 sfm) - Table 7, and Km=0.9 – Table 9.

Initial cutting speed Vc=43 × 0.9 × 1 × 1 ≈ 39 (m/min) or 128 sfm.

Spindle speed n=1000 × 39 /(π × 40)=310 (rpm).

Feed fz, which is equal to basic feed fzo in this case, is found from Table 26:

fz=fzo=0.45 mm/tooth (.0177 ipt).

The mill has 6 teeth, hence feed speed Vf=0.45 × 6 × 310=837 (mm/min) or approx. 33 ipm. Maximum depth of cut (DOC) apmax (the catalogue data) for the mill is 0.8 mm (.0031 in). Necessary ramping angle for milling lengthwise αr≈ tan-1 (apmax/(L-d))=tan-1 (0.8/180)=0.25° (**Fig. 22**), while for milling widthwise αr≈ tan-1 (apmax/(W-d))=tan-1 (0.8/110)=0.65°. Due to the maximum 0.9° ramping angle for the mill (the catalogue data), ramping in both directions with reaching the maximum DOC in the end of the pass is possible.

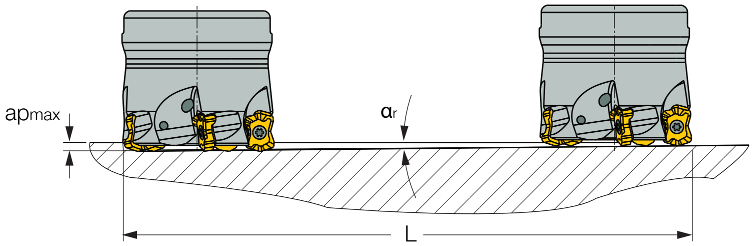


Fig. 22 Ramp-down milling

Table 27 - fzo for FF SCEM and MULTI-MASTER Heads

Family	Designation	Basic feed fzo, mm/tooth (ipt)					
		Diameter d, mm (in)					
		1 (.037)	2 (.075)	3 (.125)	4 (.160)	5 (.200)	6 (.250)
SOLIDFEEDMILL and MULTIFEEDMASTER	**EFF-S and MM EFF**	0.02	0.1	0.16	0.2	0.2	0.2
		(.0008)	(.004)	(.0063)	(.008)	(.008)	(.008)
MULTIFEEDMASTER	**MM FF**						

Family	Designation	Basic feed fzo, mm/tooth (ipt)					
		Diameter d, mm (in)					
		8 (.315)	10 (.375)	12 (.500)	16 (.625)	20 (.750)	25 (1.000)
SOLIDFEEDMILL and MULTIFEEDMASTER	EFF-S and MM EFF	0.21	0.25	0.25	0.3	0.32	0.35
		(.0083)	(.01)	(.01)	(.012)	(.126)	(.138)
MULTIFEEDMASTER	MM FF		0.37	0.37	0.45	0.48	
			(.0146)	(.0146)	(.018)	(.019)	

Example

Find cutting data for high feed milling a workpiece from titanium Ti-10V-2Fe-3Al, which will be machined by a reinforced special **MULTI-MASTER** tool carrying two-flute head MM FF120R2.0-2T08 908.

On the assumptions that the whole technological system is rigid, from Table 7 basic cutting speed Vo=53 m/min (174 sfm) Table 7, and from Table 9 Km=0.56.

Initial cutting speed Vc=53 × 0.56 × 1 × 1 ≈ 30 (m/min) or 97 sfm.
Spindle speed n=1000 × 30 / (π × 12)=795 (rpm).

Basic feed fzo is chosen from Table 27: fzo=0.37 mm/tooth (.0146 ipt).
Feed rate Vf=0.37 × 2 × 795=588 (mm/min) or 23 ipm.

Tooth structure and feed per tooth

*There are two kinds of **MULTI-MASTER** fast feed milling heads: four- and six-flute MM EFF and two-flute MM FF. The recommended feed per tooth fz for MM FF heads is 1.5 times more than for MM EFF ones. The reason is a tooth structure. If the teeth of the multi-flute heads MM EFF are produced by grinding cylindrical blanks similarly to solid carbide endmills, the teeth of "economy-type" two-flute heads MM FF are pre-shaped by sintering and then are merely finished by grinding. The high-impact structure of a sintered tooth ensures greater fz.*

High Speed Milling Titanium

High Speed Milling (HSM) Can Refer To Several Operations:

- high cutting speed milling,
- high spindle speed milling,
- high feed speed milling.

All the mentioned speeds are interrelated. Increasing spindle speed n automatically results in increasing feed speed Vf as well, and likewise more cutting speed Vc requires more spindle speed n to the same extent. As cutting speed varies in direct proportion to the diameter of a rotating tool, ensuring the same Vc for mills of different diameters demands different spindle speeds.
In the context of the guide, more correct understanding of HSM relates to high spindle speed.
In milling titanium, cutting speeds are significantly lower than in milling steel.
Although advanced tool materials and new machining techniques have led to growing a growth in averaged cutting speeds set for milling titanium, high Vc and n values for titanium may look like “normal” values for steel.

There are several factors that result in high spindle speed n:

- small tool diameter,
- small effective diameter,
- trochoidal milling as the machining strategy.

The small effective diameter de relates to milling by profiling tools (especially ball nose or lens type) at shallow depth of cut. Table 28 and **Fig. 23** show possible cases for calculating the effective diameter for ball nose milling cutters.

Example

An 8 mm (.315 in) solid carbide ball nose endmill machines the inclined surface of a titanium workpiece by ramp-up milling. Ramping angle α=12°, machining allowance per pass a=0.1 mm. The workpiece material is annealed Ti-6Al-4V, the endmill carbide grade – IC908. Find spindle speed n.

From Table 28 effective diameter
$de=(8-2\times0.1)\times\sin12^\circ+2\times\sqrt{((8\times0.1-0.1^2}\ )\times\cos12^\circ=3.36$ (mm) (.132 in).
Basic cutting speed Vo=65 m/min (213 sfm), Table 8.
For simplicity assume that cutting speed is the same (Vc=Vo).
Spindle speed $n=(1000\times65)/(\pi\times3.36)=6158$ (rpm).

Effective diameter
A profile milling tool (ball nose cutter, toroidal mill etc.) features a shaped, non-straight cutting profile and the cutting diameter is a function of a depth of cut. In profile milling, the effective diameter is the largest true cutting diameter.
Generally, it corresponds to the diameter, measured at the axial depth of cut.

A necessary cutting speed should be calculated with respect to the effective diameter.
Ignoring it can cause essential errors in cutting data and result in poor performance.

Note. For the equal cutting speed, which corresponds to nominal tool diameter d=8 mm (.315 in), spindle speed will be 2586 rpm. For the nominal diameter, the found spindle speed provides the cutting speed 154.8 m/min (507.8 sfm). Using this virtual speed as a characteristic of the considered operation is misleading because the real cutting speed is much lower.

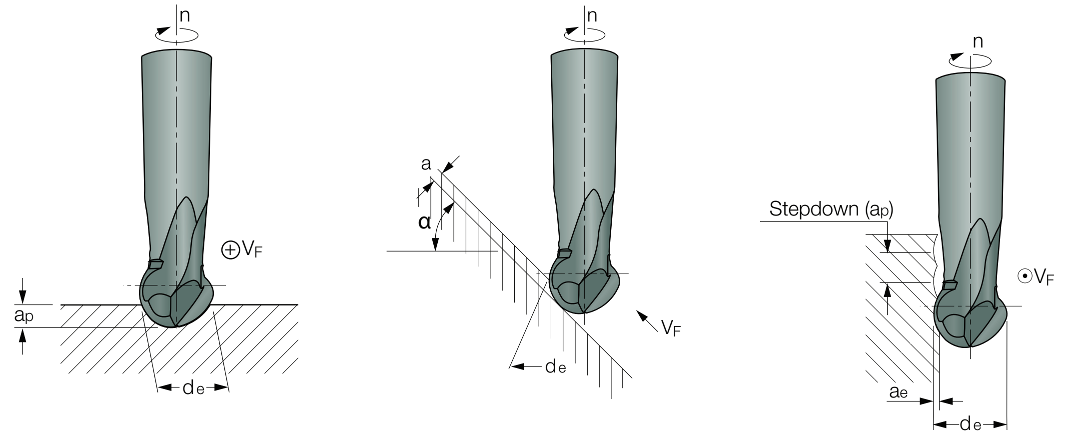


Fig. 23 Milling by ball nose cutter

Table 28 Calculating Effective Diameter De for Ball Nose Milling Cutters

Cutting path	Case in Fig. 23	Effective Diameter de	Notes
The cutter axis is normal to a machined surface	a)	$2\times \sqrt{(d\times ap-ap^2)}$	
Ramping, milling inclined surfaces	b)	$(b-2\times a)\times \sin\alpha_r+2\times \sqrt{(d\times a-a^2)\times \cos\alpha_r}$*	$\approx d\times \sin\alpha_r$**
Milling straight walls	c)	d	

* α_r - ramping angle, a - machining allowance (stock to be removed) per pass.
** The simplified equation is often used for estimating effective diameter. In many cases it gives a more or less suitable result. Nevertheless, this equation should be applied for rough calculation only due to the truncation error that can be a serious of inaccuracy.

Conventional milling of slots or grooves starts from machining solid material directly at full tool engagement. Milling with full tool engagement requires increased cutting forces and, as a consequence, consumes more power. A high speed trochoidal rough milling technique can be an effective alternative to the common slot milling strategy. In trochoidal milling, a rapidly rotated tool machines material by arc motion at a significant depth of cut and very small width of cut (usually, 5-15% of tool diameter d). The small angle of engagement AE both allows multi-flute tool design and ensures substantial increasing cutting speed Vc and feed per tooth fz. The tool slices thin layers of material with both high speed and high feed rates and this productive rough milling method features a noticeable reduction in power consumption. Trochoidal milling has been utilized successfully in manufacturing titanium parts with various slots and grooves (**Fig. 24**), and also pockets or cavities¸ especially those with relatively thin walls. The metalworking industry adopted this method for producing different integrally bladed rotors (IBR) such as blisks, blings, impellers, etc. (**Fig. 25**)

Example

An aircraft engine builder has decided to apply trochoidal milling to rough machining slots in an integrally bladed rotor. ISCAR's 10 mm (.394 in) dia. ECK-H7 7-flute solid carbide endmill, made from carbide grade IC900, will be used under the following averaged machining parameters: ap=16 mm (.63 in), ae=1 mm (.04 in). Material – Ti-6Al-4V ELI. Operational stability is good. Find initial cutting data.
Assume that Ks=1 (stable cutting).
Basic cutting speed Vo=65 m/min (213 sfm) (Table 8).
Km=1.1 (Table 9).
ae/d=1/10=0.1. Angle of engagement AE=36°, Ke=1.5 (Table 10).
From equation (1): initial cutting speed Vc=65×1.1×1.5×1=107 (m/min) or 352 sfm.
Spindle speed n=(1000×107)/(π×10)= 3406 (rpm).
Basic feed per tooth fzo=0.13 mm/tooth (0.0051 ipt) (Table 16).
Kf=0.78 (Table 17).
From equation (6): starting feed fz=0.13×1×0.78=0.1 (mm/tooth) or .004 ipt.
Feed rate Vf=0.1×7×3406=2384 (mm/min) (93.86 ipm).
Estimated metal removal rate Q=1.6×0.1×238.4=38.14 (cm^3/min) (2.36 in^3/min)

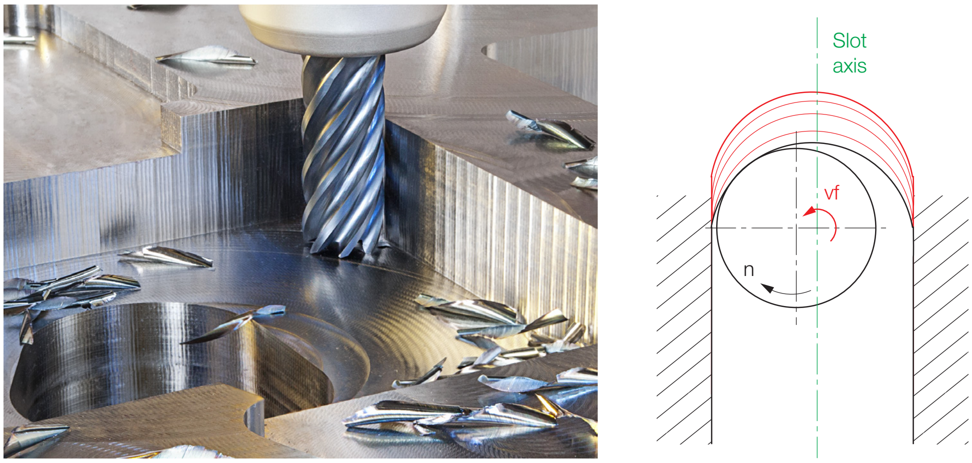


Fig. 24 Trochoidal milling

Solid carbide endmills (SCEM), which are used in machining titanium, can be successfully operated by trochoidal milling techniques and commonly feature 4-5 flutes. A variety of SCEM with greater number of flutes have been designed specifically for trochoidal milling applications. Table 29 gives basic data about ISCAR's families of these solid carbide endmills.

Table 29 SCEM Families for Trochoidal Milling Titanium

Family	Designation	Priority	Diameters D, mm (in)	Number of flutes, Z	Flute helix angle	Angular pitch	Carbide grade
Ti-TURBO	ECK-H7/9-CFR	1st choice	6-20	7 and 9*	Different for flutes	Variable	IC900
CHATTERFREE	EC-E7/H7-CF	2nd choice	2-20 (.125-.750)	7			IC902

* 7 teeth for d up to 12 mm, 9 teeth for d more than 12 mm.

Trochoid
A trochoidal curve or, simply, a trochoid, is the general name of the curves that are generated by a fixed point of a curve rolling along a directing line or curve without slip.

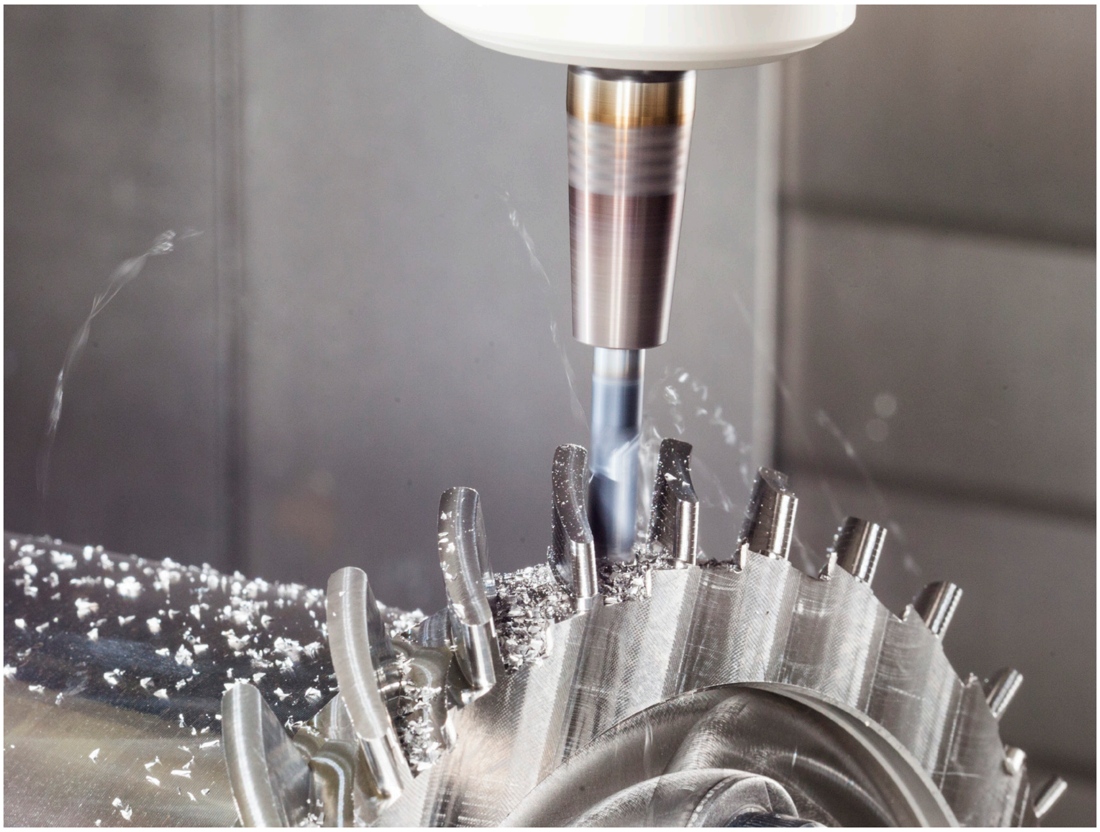


Fig. 25 Milling blisk

For optimal results of high speed trochoidal milling titanium in specific applications (milling blisk is the most typical example), manufacturers require many tailor-made (special) tools that differ in diameters, cutting and overall lengths, and a shank type. An excellent solution can be found with the use of **MULTI-MASTER** products ensuring numerous design configurations for assembled tools.

The **MULTI-MASTER** family features a rich variety of special multi-flute heads, which are intended for trochoidal milling and are produced by request.

Flying bling

A rotor, the rotating part of a turbine, mainly comprises a shaft, a disk and blades. Traditionally, the rotor is an assembly in which the blades are mounted in a disk. An alternative design features an integrally bladed rotor (IBR) - a monoblock part that integrates the blades and the disk.

In comparison with the assembled structure, the alternative reduces the rotor weight but, at the same time, makes maintenance more difficult. Depending on the specific configuration, IBR ("bladed rotor" or simply "blotor") may be considered as a bladed disk ("blisk"), a bladed ring ("bling") or even a bladed drum ("blum").

DRILLING TITANIUM

ISCAR provides different tools for drilling titanium that can be divided in two main types: solid carbide drills (**Fig. 26**) and assembled drills with exchangeable cutting parts. The latter comprise drills carrying indexable carbide inserts (**Fig. 27**) and drills with replaceable cutting heads (**Fig. 28**, 29). Assembled drills are the most common in machining titanium components. Table 30-34 show general engineering data of ISCAR's drill families that are most used in machining titanium. In drilling titanium, inner coolant supply (through the tool body) is preferable, and all the drills are suitable for this option.

Table 30 ISCAR'S Drills for Machining Titanium

Family	Designation	Type	Cutting area	Z**	Diameter range		Hole accuracy*
					mm	in	
DR-TWIST	DR	assembled	indexable inserts	1	12…60	0.469…2.344	IT12
SUMOCHAM	DCN	assembled	replaceable head	2	6…32	0.236…1.260	IT10-IT9
SOLIDDRILL	SCD ACP	solid		2	3…20	0.180…0.813	IT10-IT9

* ISO tolerance grades, provided in average cutting conditions

** number of effective cutting edges

Table 31 Cutting Length Series for Indexable Drills DR-TWIST

Diameter d*		Cutting length series			
mm	in	2×d	3×d	4×d	5×d
12 to 14	.469 to .562	▨	▨	▨	
14 up to 24	.562 up to .936	▨	▨	▨	▨
over 24 up to 38	over .936 up to 1.500	▨	▨	▨	
over 38 up to 60	over 1.500 up to 2.344	▨		▨	

* the nominal diameter of a drill.

Table 32 Cutting Length Series for Drills With Replaceable Heads SUMOCHAM

Diameter d*		Cutting length series				
mm	in	1.5×d	3×d	5×d	8×d	12×d
6 to 8	.236 to .315	▨	▨	▨	▨	
8 up to 26	.315 up to .984	▨	▨	▨	▨	▨
over 26 up to 32	over .984 up to 1.220	▨	▨	▨	▨	
	over 1.220 up to 1.260	▨		▨		

* the nominal diameter of a drill.

Table 32 Cutting Length Series for Solid Carbide SOLIDDRILL Inch Line Drills, Metric Line

Diameter d*, mm	Cutting length series		
	3×d	5×d	8×d
3 to 4.9	▨		
4.9 to 7.8	▨		▨
7.8 up to 9.9	▨	▨	▨
over 9.9 up to 12	▨	▨	
over 12 up to 20		▨	

* the nominal diameter of a drill.

Table 33 Cutting Length Series for Solid Carbide Drills SOLIDDRILL , Inch Line

Diameter d*, in	Cutting length series	
	3×d	5×d
.180 up to .343	▨	
over .343 up to .813	▨	▨

* the nominal diameter of a drill.

Fig. 26 A solid carbide drill from the SOLIDDRILL family, intended for machining titanium

No-setup time

ISCAR's tools with replaceable heads (for example ***SUMOCHAM*** *and* ***MULTI-MASTER*** *families) wholly meet the requirements of the important "no-setup time" principle, as replacing a worn head does not require additional setup operations. The head can be changed without removing a tool from a machine, which significantly decreases downtime. The* ***MULTI-MASTER*** *unified thread connection allows the shank to carry different cutting heads and vice versa, converting the shank to a universal holder and so reducing both tool inventory and storage.*

Fig. 27 DR drill with indexable inserts

Fig. 28 SUMOCHAM drills of different cutting length series

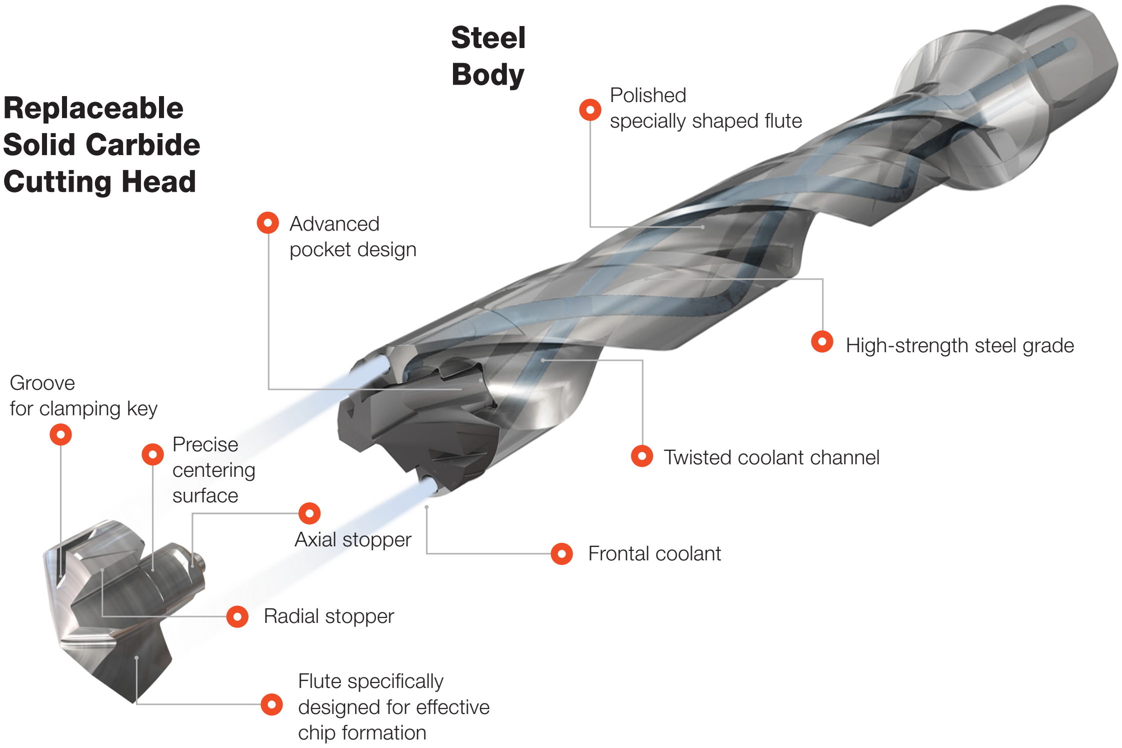


Fig. 29 The structure of a SUMOCHAM drill

Drilling Shallow Holes

Drilling shallow holes is widely used in processing, for example in short pre-drilling or spotting a hole, in order to achieve location accuracy and successful further drilling. Any drill of relatively small diameter may be applied for this operation; however, there is no need for the tool to have a significant cutting length. A better performance is achieved with the use of drills featuring stab cutting length that are intended specifically for producing short holes.

ISCAR provides various solutions for drilling shallow holes and spot drilling (Table 35) in the following design configurations: replaceable **MULTI-MASTER** heads and solid carbide drills. In addition to spot drilling, these simple tools are suitable for countersinking, milling chamfer and removing burrs.

Fig. 30 MM HCD head

Fig. 31 MM ECD head

Table 35 Spot Drills

Family	Designation	Diameter*, mm (in)	Point angle				Fig**
			60°	80°	90°	120°	
MULTI-MASTER	MM HCD	8			▨		30
		8.3			▨		
		(.375)		▨	▨		
		10	▨		▨	▨	
		1.4			▨		
		12	▨		▨	▨	
		12.4			▨		
		(.500)		▨	▨		
		(.625)		▨	▨		
		16	▨		▨	▨	
		16.5			▨		
		(.750)		▨	▨		
		20	▨		▨	▨	
	MM ECD	6			▨		31
		8			▨		
		10			▨		
		12			▨		
		16			▨		
SOLIDMILL	ECD-S2	3			▨		32
		4			▨		
		5			▨		
		6			▨		
		8			▨		
		10			▨		

* the nominal diameter of a drill.
** an illustrative example

Fig. 32 Solid carbide spot drill ECD-S

Center Drilling

MULTI-MASTER replaceable MM ECS heads (**Fig. 33**), produced from solid carbide, are intended for drilling 60° center holes in accordance with DIN 332 and ANSI B94.11M (metric and inch products correspondingly). There are two types of heads that ensure drilling the center holes of form A (without protective chamfer, "plain type") and form B (with conical protective chamfer, "bell type"). Table 36 specifies the application range for ISCAR's standard line of these heads.

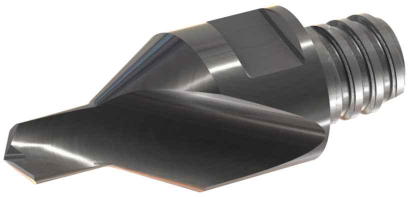
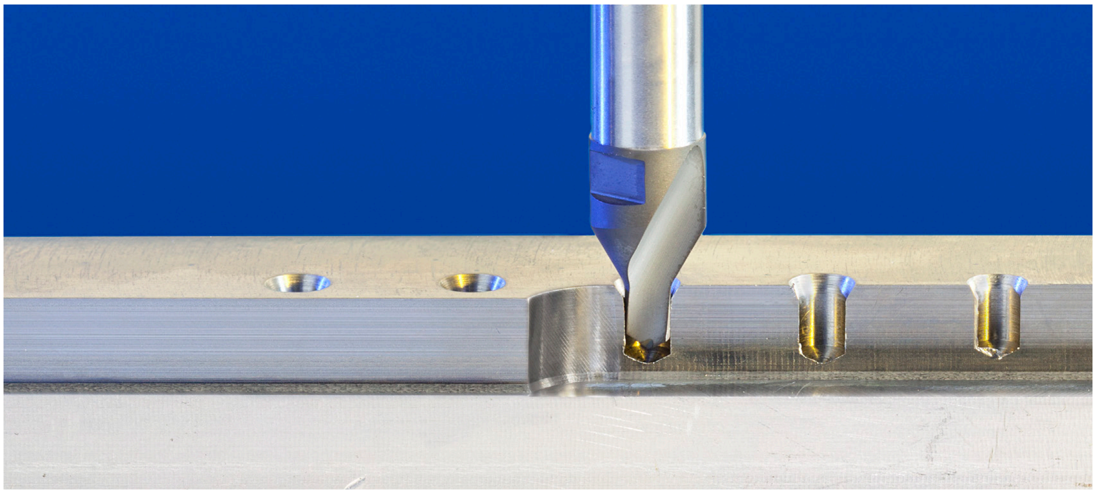


Fig. 33 MM ECS head for drilling center holes

Table 36 - Applicabillity of MM ECS Center Drilling Heads

Center hole Type	Designation acc. to DIN 332							Size acc. to ANSI B94.11M		
	1×2.12	1.6×3.35	2×4.25	3.15×6.7	4×8.5	5×10.6	6.3×13.2	4	5	6
A ("plain")	▨	▨	▨	▨	▨	▨	▨	▨	▨	▨
B ("bell")				▨	▨	▨	▨			

Drill or countersink?
A center drill is needed for forming a conical hole in workpieces. This hole is used for supporting the workpieces by the centers of machine tools. One of the methods for forming conical holes is countersinking - machining by a specially designed cutter, a countersink. In fact, the center drill performs a combination of two operations simultaneously: drilling and countersinking. Therefore, the center drill is often referenced as a "combined countersink". Other names of the center drill – "Slocombe drill" and "Slocomb drill" – are rarely used. Sometimes, the center drill is considered as a spot drill; however this specification is not strictly correct. A spot drill only drills but a center drill performs two operations: drilling and countersinking, therefore "spot a hole" and "drill a center hole" are not the same.



Selecting A Drill

Selecting a suitable drill is based on the necessary design configuration, cutting geometry, and cutting material of a tool. Hole diameter, accuracy, and required cutting length define the drill configuration, while a the geometry and the cutting material are defined by the workpiece material. **Fig. 34**, which is based on Tables 31-34, shows the application area of ISCAR's drill families intended for machining titanium, in coordinates "drill diameter-drill cutting length" in simplified form.

Central drilling: carbide vs. HSS

High-speed steel (HSS) reversible center drill bits are the most popular tools for center drilling. The bits are simple, always available for purchase, and feature low prices. This may raise the question: "Does the MM ECS replaceable solid carbide head offer a real alternative to the bits?". A seemingly obvious answer may be not so evident, especially in cases of machining difficult-to-cut material (like titanium). The carbide head ensures noticeable increases in cutting speed and feed, resulting in higher productivity and reduced machining costs. In addition, the tool life of the head is much longer. Therefore, the correct answer requires brief economical calculations, which will show the actual situation for every specific case.

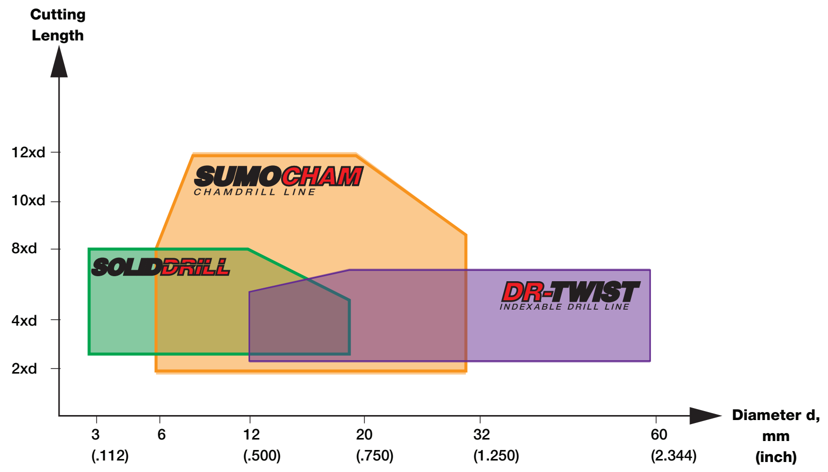


Fig. 34 Application range of ISCAR's drilling line (standard delivery)

To select the most suitable ISCAR drill for machining titanium, note the general recommendations below:

☛ ***SUMOCHAM*** *drills are the first-choice products and should be used wherever possible.*
IC908 is the preferred carbide grade.
MULTI-MASTER *tools are most recommended for spot drilling.*

Coolant

Coolant supply is a vital factor in drilling titanium. Liquid coolant is not only preferable but is essential. Although wet cooling by flooding under low pressure is still common, tool body-through (inner) coolant supply ensures high performance. Advanced drills (both solid and assembled) feature inner coolant channels for delivering coolant through the drill body, and such drills should be used whenever possible.
Recommended minimal coolant pressure is 12 bars (174 psi). Higher values will improve the results. Normally, the pressure range is 12…70 bars (174…1015 psi). Higher range values relate to machining with high pressure coolant (HPC) supply, and they require separate consideration.

Initial Cutting Data In Drilling

Cutting Speed

In drilling titanium with coolant pressure up to 70 bar,
initial cutting speed Vc is found with the use of equation (9):

$$Vc=Vo\times Km\times Ks \qquad (9)$$

Where:
Vo – basic cutting speed (Table 37)
Km – machinability factor (Table 9)
Ks – stability factor (1 for normal stability and 0.7-0.8 for unstable drilling and machining in unfavorable conditions)

Table 37 Basic Cutting Speed Vo In Drilling

Family	Designation	Vo, m/min (sfm) or drill diameter d, mm (in)					
		Up to 3 (up to .118)	Over 3 up to 6 (over .118 up to .236)	Over 6 to 12 (over .236 up to .472)	From 12 up to 20 (over .472 up to .787)	Over 20 up to 32 (over .787 up to 1.260)	Over 32 up to 60 (over 1.260 up to 2.362)
SUMOCHAM	DCN		38 (125)	45 (148)		50 (164)	
SOLIDDRILL	SCD ACP	32 (105)					
MULTI-MASTER	MM HCD, MM ECD						
SOLIDMILL	ECD-S2	32 (105)					
DR-TWIST	DR				50 (164)		

Fig. 34 ICG drilling head with chip-splitting cutting edges

Chip splitting in drilling
A chip-splitting cutting geometry may be used in drilling tools. There are different designs of drill cutting edges with chip splitting grooves, for example, ICG heads in ***SUMOCHAM*** *family (****Fig. 34****). Splitting chips into small segments improves chip evacuation and provides increased cutting speed (by an average of 15%). Under the same cutting conditions, a straight-style edge ensures better surface finish. Therefore, the chip-splitting geometry is suitable mainly for rough drilling operations.*

Feed

Following equation (10) defines starting feed f (per revolution):

$$f=fo\times Kh \qquad (10)$$

Where:
fo – basic feed (Table 38, 39)
Kh – overhang factor (Tables 40, 41)

Table 38 Basic Feed Fo for DR-TWIST DR Drills

Insert size		04	05	06	07	09	10	11	12	14
Basic feed fo	mm/rev	0.05	0.06	0.06	0.07	0.08	0.08	0.1	0.1	0.12
	inch/rev (IPR)	.002	.002	.002	.003	.003	.003	.004	.004	.005

Table 39 Basic Feed Fo for SUMOCHAM DR, SOLIDDRILL SCD ACP, MULTI-MASTER* and SOLIDMILL ECD S2 Drills

Drill diameter	mm	Up to 3	Over 3 up to 5	Over 5 up to 8	Over 8 up to 12	Over 12 up to 16	Over 16 up to 20	Over 20 up to 25	Over 25 up to 32
	inches	Up to .118	Over .118 up to .2	Over .2 up to .315	Over .315 up to .472	Over .472up to .63	Over .63 up to .787	Over .78 up to .984	Over .984 up to 1.26
Basic feed fo	mm/rev	0.04	0.07	0.09	0.12	0.15	0.19	0.21	0.23
	inch/rev (IPR)	.002	.003	.004	.005	.006	.008	.008	.009

* for tools carrying MM HCD, MM ECD and MM ECS heads

Table 40 Overhang Factor Kh for DR-TWIST DR Drills

Cutting length series	2×d*	3×d	4×d	5×d
Overhang factor Kh	1	0.93	0.82	0.75

* d - drill diameter

Table 41 Overhang Factor Kh for SUMOCHAM DR, SOLIDDRILL SCD ACP, MULTI-MASTER and SOLIDMILL ECD S2 Drills

Cutting length series*	up to 3×d**	5×d	8×d	12×d
Overhang factor Kh	1	0.87	0.82	0.80

* tool overhang for MULTI-MASTER tools
** d - drill diameter

Example

A machining division technologist plans the process for manufacturing a part, which is produced from annealed titanium Ti-6Al-4V. One of the machining operations is drilling a series of through-holes in the part base, which have thickness 65 mm (2.56 in). The holes are 24 mm (.945 in) in diameter, the diameter tolerance limits are ±0.065 mm (±.0026 in). The technologist decided to use an ISCAR drill for this operation. Recommend the drill and cutting data.

The hole depth/diameter ratio is 65/24≈2.7, so the drill cutting length series should be 3×d.
The required accuracy for the diameter relates to ISO IT11 grade. **SUMOCHAM** products are recommended as the first-choice drills. We can select the DCN 240-072-32A-3D (24 mm Dia., 3×d cutting length series) as it provides IT10-IT9 hole accuracy (Table 29).
This drill carries the head ICM 240 IC908 (IC908 – the preferred carbide grade).

Basic cutting speed Vo=38 m/min (125 sfm), Table 37. Machinability factor Km=1 (Table 9).
By assuming that the operational stability is normal (Ks=1),
equation (9) gives starting speed Vc as below:
Vc=38×1×1=38 (m/min) or 125 sfm.
Spindle speed n=(1000×38)⁄(π×24) =504 (rpm)

Basic feed fo=0.21 mm/rev (0.0082 IPR), Table 39.
Overhang ratio Kh=1 (Table 41).
Calculating feed f with the use of equation (10) gives the following result:
f=0.21×1=0.21 (mm/rev) or 0.0082 IPR.
Feed speed Vf=0.21×504=105.8 (mm/min) or 4.13 IPM.

Short and deep holes
Cutting length series
Commonly used terms “short” and “deep” holes do not have a strict definition.
It is widely accepted that drilling a hole of diameter d and (10…12)×d or higher in depth relates to deep drilling, while holes having depth up to 5×d, are short.
In the terminology used by ISCAR, only a drilling depth of 12×d and higher is considered as deep. Consequently, the holes with shallower depths are short. The drills intended for machining short holes vary in their cutting length series: short (up to 3×d), long (4×d and 5×d) and extra-long (8×d and 12×d).

REAMING TITANIUM

Reaming is not a very common operation in machining titanium; however, various processes require it. ISCAR's line of reaming tools comprises several families of reamers. The families differ in their design: a solid carbide concept or an assembled one. Reaming titanium features a relatively large proportion of required non-standard dimensions for the nominal diameter of a hole and its tolerance limits. It often is caused by "springiness" of titanium and this characteristic should be considered during process planning. ISCAR provides tools for reaming titanium at the customer's request, as tailor-made products; the most suitable solutions are customized high-speed reamers on the base of the **BAYO T-REAM** family (**Fig. 35**).

A typical **BAYO T-REAM** reamer (**Fig. 36**) is an assembled tool comprising a holder (also referenced as a shank) and an interchangeable multi-flute reaming head, produced from cemented carbide. The head is mounted in the holder with the use of a unique quick-change bayonet mechanism. The heads feature different connection size ("bayonet number") that has designation BN5, BN6, …, BN9. Each size corresponds to a specific area within the diameter range of the reamers. There are heads with different flute directions depending on the type of a machined hole: through or blind (**Fig. 37**). The holder material is usually steel but, if necessary, it may be also heavy metal and cemented carbide. The reamers are suitable for inner coolant supply where the coolant flow is directed on every cutting edge.

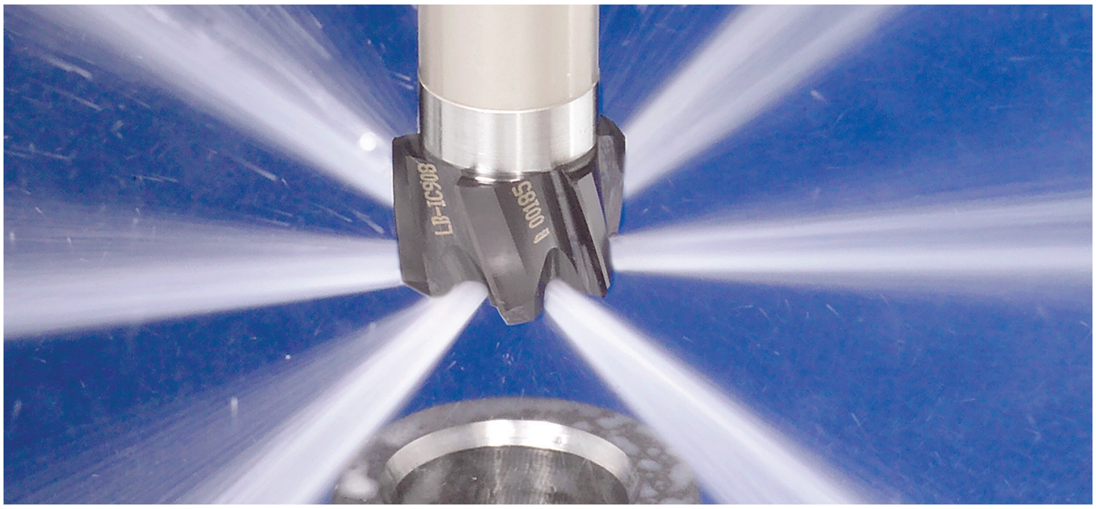


Fig. 35 A BAYO T-REAM reamer

The design concept of the family offers the following advantages:

1. Versatility – the holder can carry different heads, and the head is suitable for mounting in different holders. This feature ensures a wide usage of standard holders, diminishes needs for tailor-made ones, and thus reduces tool stock.
2. "No setup time" principle – replacing a worn head by a new one does not require additional setup operations and can be done when a reamer is clamped directly in the spindle of a machine tool. It is an efficient means to cut the machine downtown and reduce production cost.
3. Easy-to-use – the head is replaced in a quick and simple way.

Table 42 contains general data that characterize the tools for reaming titanium from the ***BAYOT-REAM*** family

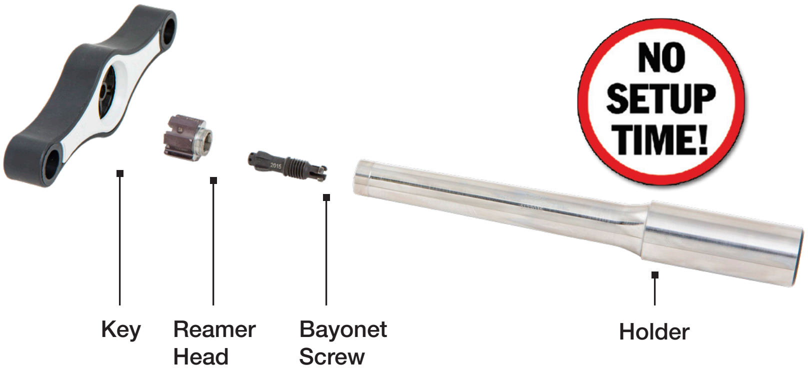


Fig. 36 Elements of a BAYO T-REAM reamer

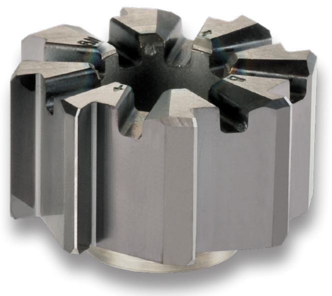


Fig. 37 An interchangeable reaming head with straight flutes

Table 42 BAYO T-REAM Tools for Reaming Titanium: General Data

Parameter	Value in units	
	Metric	US customary
Range of diameters	11.5…32 mm	.4528…1.26 in
Hole accuracy*, ISO grade	IT 7	
Hole surface finish*, Ra	0.2…0.4 µm	8…16 µin

* in average cutting conditions

Initial Cutting Data In Reaming

Reaming Allowance

When reaming titanium, guide values for the reaming allowance – a material stock (per diameter) left for reaming – are shown in Table 43.

Table 43 Diametral Reaming Allowance for BAYO T-REAM Tools

Allowance for hole diameter		
From 11.5 to 13.5 (from .4528 to 05315)	From 13.5 to 16 (from .5315 to .63)	From 16 to 32 (from .63 to 1.26)
0.15-0.25 (.006-.01)	0.20-0.30 (.008-.012)	0.20-0.35 (.008-.014)

* units: mm (in)

Cutting Speed

Initial cutting speed Vc is calculated by equation (11):

$$Vc=Vo\times Km \quad (11)$$

Where: Vo=45 m/min (148 sfm) – basic cutting speed
Km – machinability factor (Table 9)

Feed

Table 44 provides data for setting starting feed per tooth fz.

Table 44 Recommended Feed Values for BAYO T-REAM Tools*

Connection Size	Feed fz, mm/tooth (IPT)			
	Through hole		Blind hole	
	Smooth	Interrupted	Smooth	Interrupted
BN5…BN6	0.04-0.05-0.07 (.0016-.002-.0027)	0.03-0.04-0.06 (.0012-.0016-.0024)	0.03-0.04-0.06 (.0012-.0016-.0024)	0.03-0.03-0.05 (.0012-.0012-.002)
BN7…BN9	0.05-0.07-0.09 (.002-.0027-.0035)	0.04-0.06-0.08 (.0016-.0024-.0031)	0.04-0.06-0.07 (.0016-.0024-.0027)	0.04-0.05-0.06 (.0016-.002-.0024)

* recommended feed values are given as: minimal-starting-maximal

☛ *When reaming with **BAYO T-REAM** tools, the best results are reached with coolant pressure 15-20 bars (217-290 psi).*

Example

Find cutting data for reaming a through hole in the process of manufacturing a titanium part. The titanium grade is Ti-10V-2Fe-3Al, the required hole diameter is specified as Ø25Js8. It is planned to apply a **BAYO T-REAM** reamer carrying a specially designed 8-flute head with BN8 connection.

Machinability factor Km=0.56 (Table 9).
According to equation (11), initial cutting speed Vc will be as follows:
Vc=45×0.56=25 (m/min) or 83 sfm.
Spindle speed n=(1000×25)/(π×25)=318 (rpm)

From Table 44 starting feed fz=0.07 mm/tooth (.0027 IPT).
Feed speed Vf=0.07×8×318=178 (mm/min) or 7 IPM.

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

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

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

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

Belarus
JV ALC “TWING-M”
Tel +375 17 506-32-38
+375 17 506-33-31/65
Tel/Fax +375 17 506-32-37
info@twing.by
www.twing.by, www.iscar.by

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

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

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

Chile
SANDE SA
Tel +56 2 695 1700
Fax +56 2 697 0332
logistica@sande.cl

China
ISCAR CHINA
Tel +86 10 6561 0261/2/3
Fax+86 10 6561 0264
iscar@iscar.com.cn
www.iscar.com.cn

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

Croatia
ISCAR ALATI d.o.o.
Tel +385 (0) 1 33 23 301
Fax +385 (0) 1 33 76 145
iscar@zg.t-com.hr
www.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
www.iscar.cz

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

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

Germany
ISCAR GERMANY GmbH
Tel +49 (0) 72 43 9908-0
Fax+49 (0) 72 43 9908-93
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
www.iscar.hu

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

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

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

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

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

Latvia
SIA EKL/LS
Tel +371 6 733 11 54
Fax +371 6 780 56 48
ekllpstools@isr.lv

Lithuania
MECHA, UB
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
www.iscar.com.mx

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

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

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

Russia
Moscow
ISCAR LLC
Tel/fax +7 495 660 91 25/31
iscar@iscar.ru
www.iscar.ru

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
www.iscar.sk

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

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

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

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

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

Taiwan
ISCAR TAIWAN LTD.
Tel +886 (0)4-24731573
Fax +886 (0)4-24731530
iscar.taiwan@msa.hinet.net
www.iscar.org.tw

Thailand
ISCAR THAILAND LTD.
Tel +66 (2) 7136633-8
Fax+66 (2) 7136632
iscar@iscarthailand.com
www.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
www.iscar.com.tr

Ukraine
ISCAR UKRAINE LLC
Tel/fax +38 (044) 503-07-08
iscar_ua@iscar.com
www.iscar.com.ua

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

United States
ISCAR METALS INC.
Tel +1 817 258 3200
Tech Tel 1-877-BY-ISCAR
Fax+1 817 258 3221
info@iscarmetals.com
www.iscarmetals.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.iscarvn.com

Please check ISCAR’s online catalog www.iscar.com for the most current technical information regarding our products.

The ISCAR Reference Guide for Machining Titanium

Member IMC Group

www.iscar.com