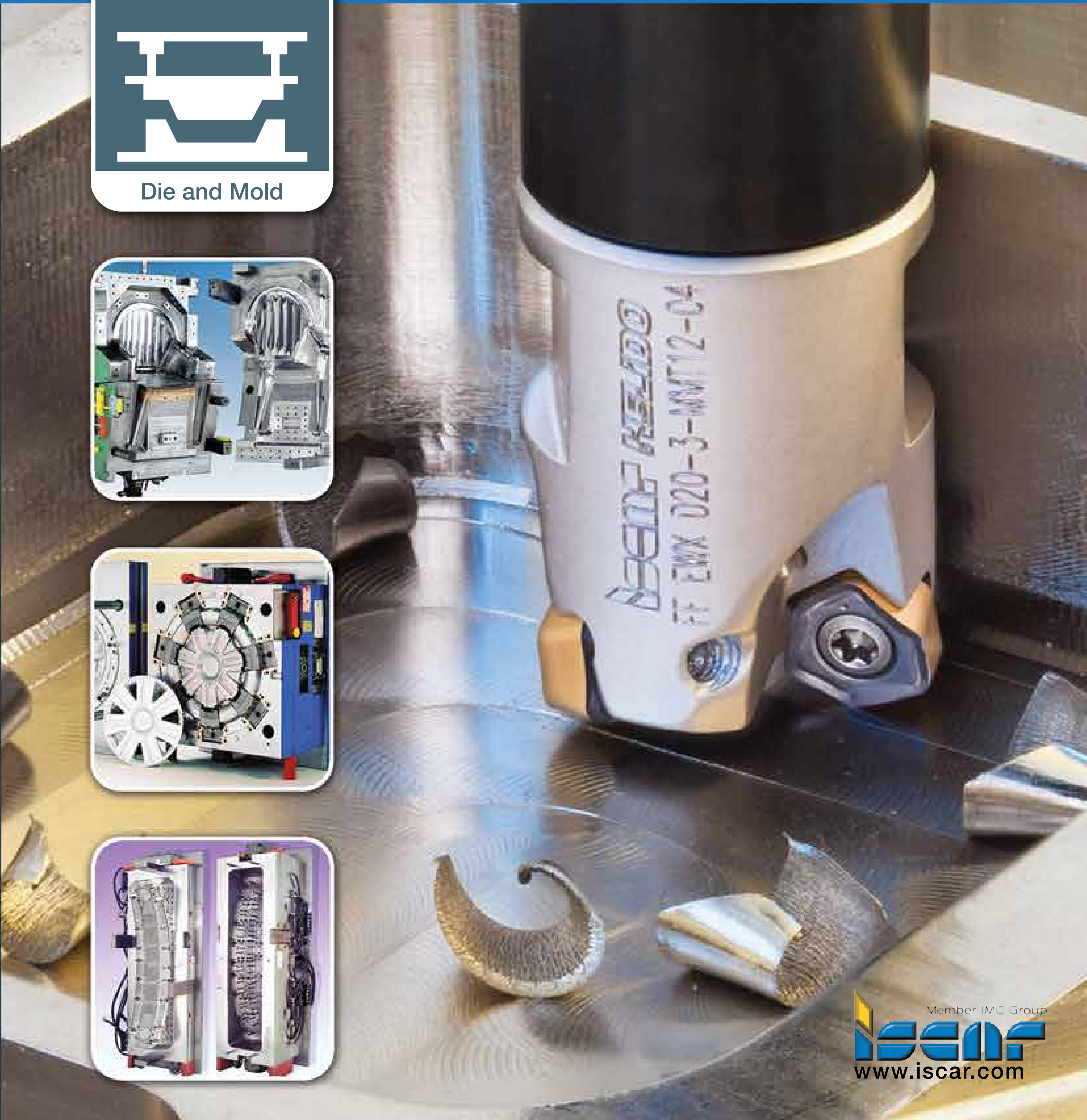
ISCAR's Quick Reference Guide for Die and Mold Making

Metric Version

Third Edition

Die and Mold



Dear Die and Mold Maker,
our Distinguished Customer and Colleague,

Our primary concern, as a tool manufacturer, is to provide you with the most progressive and most high-efficiency cutting tools that will meet your requirements and answer the purpose of modern technology.

Cutting metal faster and more accurate means cutting machining time and cost per part.
In many cases the cutting tool, which is sometimes seen as the more insignificant link in the chain of production costs, presents a barrier to achieving productivity. We, at ISCAR, are aware of that fact; and our ongoing research and development intends to develop those cutting tools that will increase your manufacturing productivity, improve the performance of your workshop and lead your die-making process to be more profitable.

The research and development result in various innovative solutions that we offer to our customers. The variety of ISCAR tools is very rich, and sometimes, it is not so simple to be well oriented in it. Therefore, we hope that this guide will help you in right tool selection and will be a good supplement to our catalogs, reference forms and leaflets because it takes into account specific features of die and mold making. First of all the guide emphasizes the latest solutions in order to give you the opportunity of becoming familiar with them.

Also, we included here some more general information, tips and even historical issues related to the discussed material and hope that you will find them useful.

We will be thankful for your every remark or suggestion regarding the guide.

We consider ourselves as your true colleague in the die and mold making process at your location and will be proud and sincerely pleased if you will feel the same.

Member IMC Group

Contents

ISCAR Tool Families Referenced in this Guide

1. Milling Tools



BALLPLUS
CHATTERFREE SOLID MILL LINE
DROPMILL
FEEDMILL
FINISHRED
FLEXFIT

HELIBALL
HELIDO 490 LINE
HELIDO 600 UPFEED LINE
HELIDO 845 LINE
HELIDO ROUND H400 LINE
HELITANG T490 LINE

MILLSHRED
MULTI-MASTER
SOLIDMILL SOLID CARBIDE LINE
SUMOMILL 290 LINE
TANGPLUNGE PLUNGING LINE

2. Hole Making Tools



BAYO T-REAM
CHAMGUN
DR-TWIST INDEXABLE DRILL LINE
ISCARDR-DH

SOLIDDRILL
SUMOCHAM CHAMDRILL LINE
SUMODRILL

Foreword

A lot of things around us, parts or even fully completed products, are produced in dies and molds: an internal cylinder block or a toy, a plastic container or a crankshaft, a bottle or a jet turbine blade, a tin soldier or a boat. These objects differ in their form, material, sizes, mechanical properties, and are manufactured by different technological methods of metal forming or processing of plastics.

Broadly speaking a die or a mold, being an assembly unit, comprises various elements, and many of them are standard or unified. A form of a part that is produced in the die or the mold is determined by several main components (cavities, rams, etc.).
Shaping the components is a central operation in die and mold manufacturing that demands from die and mold makers rich knowledge, skills and experience. A die and mold maker is rightfully considered as one of the most professionally skilled workers in manufacturing. Modern **CNC** technology and **CAD/CAM** systems substantially change die and mold making, turning it from craft into a whole branch of trade.

A part intended to be produced in a die or mold dictates shape and sizes of the die or mold and accuracy requirements; and a forming technology and the part run size – the die or mold material. The part shape and the die or mold material are a source data for die and mold making.

Automotive Industry is a major consumer of dies and molds. Approximately 60% of stamping dies and 40% of plastic molds produce automotive parts.

There are different kinds of dies and molds, which can be grouped in the following principal types: forging dies, stamping (pressing) dies, die casting dies and molds for plastics.

In manufacturing a die or mold, the shape and the sizes are the main factors of the degree of difficulty for machining:

- Low degree in case of plane areas, simple shapes, shallow cavities etc.
- Medium, when the shape becomes more complicated, the sizes bigger and the cavities deeper; rams have steep walls, etc.
- High degree in the context of very complex shape, narrow and deep cavities, considerable difference between heights, etc.

Also, difficulty in machining is a function of machinability of the die or mold material.

The key to productive and effective die and mold making is the process planning – the choice of technology of the die or mold manufacturing that, in general, includes machining operations, assembly and finishing works. Today's **CAD/CAM** software enables analyzing the die or mold design, defining machining strategy, developing **CNC** programs and machining simulation in order to find the most efficient solution that allows full use of advantages of modern machine tools.
The right machining strategy is directly related to correctly chosen cutting tools that perform material removal during operation. A tool, which seems to be a secondary element of the die and mold process, is a substantial factor of productivity and profitability.
We, at **ISCAR**, distinctly understand the role of cutting tools in the die and mold industry and try to provide the die and mold maker with reliable and efficient tooling that meets every requirement of the branch. The right tool selection depends on different factors.
We will discuss them here briefly, explain the tools' features and thus build a base for the correct tool choice.

Die and Mold Materials

We have already noticed that a die or mold has different movable and stationary parts (clamping elements, springs, pins, bolts, bushings, support pillars, etc.) generally made from different engineering materials, from plastics to cemented carbides.

However, key die or mold parts that act on specific requirements of the die and mold industry usually are from particular materials, which should be emphasized.

Tool Steels

Tool steels relate to a type of steels that, as indicated in its name, is intended first of all for making tools for cutting and forming metals and other materials. There are many national and international standards for specified tool steels. Moreover, in order to answer to the particular requirements of industry, steel manufacturers produce different steels in accordance with their own specification. These steels often have no standard designation and are identified by their trade names. In this guide we use the standards of the **American Iron and Steel Institute (AISI) and the Society of Automotive Engineers (SAE).** The parallel designation in conformity with other acting standards is sometimes given in the text; and the information section at the end of this guide contains a cross-reference table with comparative designation of tool steels in keeping with the different national standards.

Die and mold makers deal with steel in wide ranges of hardness, from low (HB 200 and less) to high (HRC 63).

In line with the main field of application there are six general and two special-purpose classes of tool steels, from which the following are the most popular in the die and mold industry:

- Cold-work tool steels including A series (air-hardening medium-alloy), **D** series (high-carbon high-chromium) and **O** series (oil hardening)
- Hot-work **H** series
- Water hardening **W** series
- Plastic mold **P** series
- Shock resistant **S** series
- Special-purpose **L** series (low-alloy)

In die and mold design, the main properties of tool steels are strength, wear resistance, corrosion resistance, etc. However, while for a die and tool maker dealing with the material, which has already been specified by designers, more important properties are: hardness, machinability, polishability and dimensional stability.

The steel manufacturers supply steels in different delivery conditions: annealed, pre-hardened and hardened. Consequently, in die and mold making process, hardness of tool steel (even the same grade) can vary within a wide range from HB 200 and less (soft steel) to HRC 63 (hard steel). Normally, high stock removal rate characterizes rough machining of a soft material while closed allowances are usual for finish cuts when material hardness is high. The term "pre-hardened steel" is not well-defined. It means that steel is hardened and tempered to relatively not high hardness but different steel producers use different limits for its specification. Generally, it is less than HRC 45, however, in technical literature and references the steels with that hardness often relate to hard steel. The term and its hardness limit are allied to cutting tool development and their ability to cut material. Therefore, steels can be divided into the following conditional groups depending on their hardness:

- Soft annealed to hardness up to HB 250
- Pre-hardened to two ranges:
  - HRC 30-37
  - HRC 38-44
- Hardened to two ranges:
  - HRC 45-49
  - HRC 50-55
  - HRC 56-63 and more

Table 1 shows some features of the most typical tool steels that are common in the die and mold industry.

Table 1 **Typical Tool Steels for Die and Mold Making**

Category	Designation		Delivery		Hardening	Application examples
	AISI/SAE	DIN W.-Nr.	Annealed to	Prehardened		
Cold-work tool steels	A2	1.2363	HB 220		HRC 56-60	Cold blanking, extrusion, coining dies, molds
	D2	1.2379	HB 210		HRC 56-62	Cold stamping and extrusion dies and punches, forging dies, master hobs
	D3	1.2080	HB 240		HRC 56-62	Cold stamping dies and punches
	O1	1.2510	HB 200		HRC 58-62	Forming dies and parts, cold stamping dies
Hot-work tool steels	H11	1.2343	HB 180		HRC 46-52	Hot extrusion dies, plastic molds
	H13	1.2344	HB 190		HRC 44-54	Die-casting dies, punches, plastic molds
Plastic mold steels	P20	1.2330	HB 280	HRC 32-36	HRC 48-52	Plastic molds
Shock-resistant steels	S7		HB 200		HRC 50-58	Hot forging dies, punches, master hobs, cold extrusion dies
Special-purpose steels	L6	1.2713	HB 230	HRC 36-44	HRC 50-60	Drop forging dies, die-casting dies, plastic molds

Not relating directly to tool steels, so called **maraging steels** (the term includes the words *martensitic* and *aging*) became be more and more usable in die and mold making, particularly, in manufacturing plastic molds. They have relatively high Ni and Co percentage, can be supplied in pre-hardened state (HRC 30-36, Maraging 250 and Maraging 300, for instance) and are hardened to HRC 58. Nevertheless, despite their dimensional stability and good polishability, they are still considerably expensive comparing with mold steels, and therefore their application in the die and mold industry is limited.

Stainless Steels

Martensitic stainless steels (AISI 420, 420F) are widely used for producing plastic molds, especially, for cavities of small to medium sizes with complicated shapes and substantial differences in cross-sections. They are supplied as annealed to HB 200, and their heat treatment after machining is simple.

Alloy Steels and Plain Carbon Steels

Alloy and even plain carbon steels are used for various general parts such as holders, pillars, pins, rings, etc. Moreover, in many cases (low-run production dies, some kinds of molds) AISI/SAE 4130, 4140 and 4150 steels are main materials for forming parts. These three steels are usually supplied in an annealed state or pre-hardened to HRC 32-35 and can be hardened to HRC 44-52.

The die and mold industry handles practically all types of engineering materials, but the most typical for the branch are tool steels.

Cast Iron

Cast iron (especially grey) is also considered as a die and mold material for manufacturing large-sized parts, plates, spacers, bushings and other components where wear is not expected. In addition, nodular cast iron sometimes is used for dies, punches, jigs and pads and even for molds.

Nonferrous Metals

Aluminum is not the most popular material in the die and mold industry, but it often used for prototype dies and molds, for multiplied identical molds, short life molds and various extrusion molds because it is easy to machine and low cost. Today aluminum starts to penetrate into resin mold manufacturing due to its much better thermal conductivity, machinability and polishability compared to mold steels. The following aluminum alloys: 2024, 6061 and 7075 (in accordance with the Aluminum Association Alloy and Temper Designation System) are more and more common in mold making practice.

Beryllium-copper alloys and zinc alloy are materials for blow molds, injection mold components and cavity inserts. Modern metal producers offer beryllium-copper alloys that have enough strength and good wear and corrosion resistance properties. In machining, they are cut 2-3 times faster than tool steels. The alloys' hardness is HRC 30-42, depending on the hardness grade. For this reason the beryllium-copper alloys can replace traditional tool steels and stainless steels as a mold material in some cases.

Electrical discharge machining (EDM) is another important field of nonferrous metals application in die and mold making. Electrodes for EDM are produced from brass, copper, copper tungsten (60-70% of tungsten) and graphite.

Machinability of Die and Mold Materials

The following data can be useful for estimating machinability of common die and mold materials in regular delivery conditions. Machinability rating is based on AISI/SAE 1212 free cutting steel as 100%.

Carbon steels

1020	72%	1030	72%	1045	57%
1060	51%	1212	100%	1215	136%

Alloy steels

4130	72%	4140	66%	4340	57%
5015	78%	8620	68%	8720	68%

Tool steels

A2	42%	D2	27%	D3	28%
D7	25%	H11	49	H12	46%
H13	46%	H19	43%	H21	36%
L6	39%	O1	42%	O6	57%
P2	42%	P5	42%	P20	38%
P21	38%	S1	36%	S5	31%
S7	45%	W1	48%	W2	45%

Maraging 300____________33%

Martensitic stainless steels

403	55%	420	45%	430	45%

Grey cast iron (ASTM A48-76 classes[1])

20 (DIN GG 10)	73%	40 (DIN GG 25)	48%

Nodular cast iron (ASTM A 536-80 classes)

65-45-12 (DIN GGG 50)	61%	80-55-06 (DIN GGG 60)	39%

Aluminum alloys

6061-T	200%	7075-T	140%

Copper and copper alloys____________80%-120%

[1] ASTM is the American Society for Testing and Materials

Typical Examples of Dies and Molds

Forging die

Die component	cavity
Material	S7
Hardness	HRC 52
Machining difficulty	medium to high
Run size	high-run production
Formed part	connecting rod

Stamping die

Die component	punch
Material	D2
Hardness	HRC 60
Difficulty for machining	medium
Run size	high-run production
Formed part	hood

Die-casting die

Die component	cavity
Material	H21
Hardness	HRC 51
Difficulty for machining	low to medium
Run size	medium-run production
Formed part	tap's housing

Die-casting die

Die component	cavity
Material	H13
Hardness	HRC 54
Difficulty for machining	high
Run size	high-run production
Formed part	crankshaft

Plastic mold

Mold component	cavity
Material	H13
Hardness	HRC 50
Difficulty for machining	low
Run size	medium-run production
Formed part	cellular phone housing

Plastic mold

Mold component	cavity
Material	P20
Hardness	HRC 48
Difficulty for machining	medium
Run size	high-run production
Formed part	car bumper

Cutting Tools in Die and Mold Industry

In die and mold manufacturing, there are different machining operations: cutting (milling, drilling, reaming, etc.), abrasive machining (grinding, polishing, honing, etc.) and EDM. Even water jet cutting is used by die and mold makers. However, metal cutting remains to be the predominant method of die and mold production.

Dies and molds have different shapes and sizes, varying from small to large. In many cases machining dies and molds requires removing a large amount of material. A typical machining process contains rough and finish cutting operations. The main parameter for rough machining with a large stock allowance is metal removal rate, while for finishing, the most important factors are accuracy and surface finish.

For the development of machine tools, CNC control and CAD/CAM systems cardinally changed the die and mold industry by giving the die and mold maker new methods of multi-axis machining and introducing advanced computer techniques of machining simulation and verification. Modern cutting strategies, such as high speed machining (HSM), high feed milling (HFM) and trochoidal milling have penetrated into die and mold production. Machining hard materials, long tool life, stability, reliability and high performance intended for reducing or even full elimination EDM and manual polishing – only the tool that meets these requirements can be considered a passport to success in productive and effective die and mold making.

Milling Tools

Milling plays a key role in machining dies and molds. Indeed, milling tools remove the most share of material, shaping a workpiece to a die or mold part. A conventional process planning comprises rough, semi-finish and finish milling operations. The traditional approach to rough milling is based on cutting with large depth and width of cut. Correspondingly, it demands high-power machine tools with low spindle speeds for large milling tools. This way of machining provides maximum productivity when cutting a soft material. Due to a die or mold has appropriate hardness requirements and in addition, heavy-duty rough milling leads to significant residual stresses, further heat treatment is necessary. The mentioned approach usually characterizes production of large-sized die and mold parts that have considerable differences in depth or height. However, some producers are still supporters of this method due to limitations of available machine tools, CNC programs or traditional thinking in process planning. Over time, and as industrial innovations developed, HFM became a relevant technique in roughing. It allows machining soft to pre-hardened steels with small depth of cut and extremely high feed per tooth and leads to increased productivity.

HSM, another modern way of metal cutting, is intended first of all for finish milling of hard steels. Nevertheless, it can be effective also for rough and semi-finish machining, particularly for small to medium parts or in cases with slight differences in depth or height because it enables cutting hard material directly. Further HSM development has resulted in trochoidal milling.

Yet one method of rough milling growing in popularity in the die and mold industry is plunge milling (or plunging) with a tool feed direction towards the tool axis. It gives the opportunity for efficient roughing of cavities and external surfaces with a complex shape (so called sculpturing operations).

The modern milling techniques, their advantages and problematic points and requirements of cutting tools will be discussed on the following pages of the guide.

All typical milling operations are involved in die and mold manufacturing: 90° shoulder milling, face milling, milling slots, contours and chamfers; and profile milling (the parallel definition of the operations: shouldering, facing, slotting, countering, chamfering and profiling, also are often used by professionals). The latter, including machining shaped 3D surfaces, is the pivot of die and mold making. Milling tools are available in different configurations: indexable, that has replaceable cutting inserts or whole cutting heads, and solid.

Selection Guideline: How to Choose the Right Milling Tool

Putting the question of tool selection into broad perspective, the main side of the issue shall be emphasized: cost per unit (CPU) for a part that is machined by the tool. In spite of the fact that the tooling cost share in CPU is minor, the tool's indirect influence on CPU reduction can be considerable. Namely the tool, this small part of a manufacturing process, sometimes is a single obstacle for a machine tool to run faster and thus to cut machining time.
Hence, for better productivity and as a result for lower CPU the most high-efficiency tool should be used.
Another important aspect is versatility of the cutting tool, its ability to perform various milling operations effectively. For example: shoulder milling, ramping and plunge milling. Such combinations allow for using one tool for different applications and, when machining a part, shortening time needed for tool change during machining.
An additional way of increasing versatility is using the tools with interchangeable precise cutting heads, which render a possible head change when the tool or its holder is clamped into a machine spindle and does not require time for setup procedures.

Taking these so obvious, but often left out points into consideration and speaking about the tool selection more specifically, the analysis by chain: Application-Geometry-Grade (AGG) shall be applied. In brief, AGG means the following commonly known checkpoints - questions, answer on which allows for the tool choice:

Application	What is the type of machining operation?
	Workpiece: its material, hardness before the operation
	Required accuracy and surface finish
	Machining allowance
	Machining strategy
	Type of machining (light, medium, heavy)
	What type of tool, in accordance to adaptation (a mill with shank, shell mill)?
	Operation stability (good, bad)
	Machine tool (sufficient/limited power, condition, spindle speed)
	Coolant (coolant type: dry, wet; possibility of coolant throw spindle)
Geometry	Which cutting geometry is recommended for machining the workpiece for the above requirements? Both types of tools should be checked: indexable and solid.
Grade	Which grade of a cutting tool material is more suitable for machining the workpiece for the above requirements?

Carbide Grades for Indexable Milling Inserts

The indexable inserts for milling the die and mold materials are produced from different tungsten carbide grades, mostly coated by methods of physical or chemical vapor deposition (PVD and CVD respectively).
PVD coatings have a wide distribution in milling inserts and solid carbide endmills because they leave the cutting edges sharp. PVD coatings are applied at relatively low temperature (about 500 ˚C).
CVD coatings are much thicker and that contributes to wear resistance. The CVD coatings are applied at high temperature (approximately 1000 ˚C). Current technology enables improving CVD process by moderate-temperature CVD (MT CVD) with its lower deposition temperatures. Further steps in technology development include combinations of both coating methods: CVD and PVD.
ISCAR offers a rich program of the carbide grades for the milling inserts. We observe briefly the more recent of them intended primarily for machining the popular die and mold materials. Except for grade DT7150, the preferred grades, which are produced by SUMO TEC method, have

a special post-coating treatment with extra advantages. Table 2 contains the comparative data for general characteristics of the grades.

IC808 is a grade with tough submicron substrate and titanium aluminum nitride (TiAlN) PVD coating, designed for milling die and mold materials such as hard alloy and carbon steels at medium-to-high cutting speed. They are noted for excellent notch wear and built-up edge resistance.

IC5100 – a tough substrate with a MT CVD and alpha aluminum oxide (Al_2O_3) coating that is recommended for milling grey cast iron at high cutting speeds, providing excellent tool life.

IC810 – a PVD grade coated with aluminum titanium nitride (AlTiN) that produces high oxidation resistance, enabling machining at high speed. A good choice for miling nodular and grey cast iron at low to medium cutting speed under unstable conditions.

DT7150 – a carbide grade with a tough substrate that has a dual MT CVD and TiAlN PVD coating. Features high wear and chipping resistance. Recommended for medium to high cutting speeds for machining of cast iron and cast steel, especially with siliceous skin.

IC830 – a PVD TiAlN coated tough grade for milling alloy steel and stainless steel. Shows good results for interrupted cut and heavy duty operations.

IC330 – a multi-purpose tough grade with titanium nitride (TiN)/titanium carbonitride (TiCN) coating. Used for milling a wide range of die and mold materials, at low to medium cutting speed.

SUMO TEC carbide grades

The SUMO TEC grades feature a special post-coating treatment which provides substantially improved life and better reliability. The new process enhances toughness and chipping resistance, reduces friction and built-up edge, thus increasing tool life. The golden-colored flank facilitates wear detection. The post-coating treatment has the effect of making the rake face even and uniform, minimizing inner stresses and droplets in coating, which lead to smooth chip flow and extended tool life. Yet at the same time the untreated flank continues to be rough enough for good contact with base surfaces of tool pockets.

Table 2 **Selected Carbide Grades for Indexable Inserts Intended for Milling Die and Mold Materials**

ISO Class DIN/ISO 513	Harder ◄ Carbide Grades ► Tougher					
	IC808	**IC5100**	**IC810**	**DT7150**	**IC830**	**IC330**
P	P15-P30	P10-P25	P15-P30		P20-P50	P25-P50
M	M20-M30				M20-M40	M30-M40
K	K20-K40	K05-K20	K05-K25	K05-K25	K15-K40	
H	H20-H30					
Coating	PVD	MT CVD	PVD	Combination	PVD	PVD

☐ - First choice

Ferritic and martensitic s.s. (ISCAR material groups 12;13)

For more detailed information about the carbide grades and coating technology refer to ISCAR catalogs, guides and technical leaflets.

1 90° Shoulder Milling

ISCAR endmills that are intended mostly for 90° (square) shoulder milling but used also for milling slots and grooves, are being developed in the following directions:

- Endmills with indexable inserts of diameter range 8-50 mm
- Solid carbide endmills with diameters to 25 mm
- Replaceable milling heads with **MULTI-MASTER** adaptation (or simply **MULTI-MASTER** heads) of 6-25 mm diameters
- Extended flute endmill cutters of diameter range 12-100 mm

The extended flute endmill cutters are for roughing, while the other kinds of endmills are used in all kinds of operations: roughing, semi-finishing and finishing.

Insert with helical cutting edge

In the 1990's ISCAR introduced new milling cutters which carried indexable sintered carbide inserts APKT 1003 PDR-HM. It was the first insert with a helical cutting edge. Now an insert cutting edge as a part of a helix that is built on a cutting cylinder seems apparent, and many tool manufacturers use this approach for producing their milling inserts.
But in the 1990's it was a real innovative solution. A helical edge ensured constant cutting geometry, smooth cutting and considerably improved insert life. Ground HSS and solid carbide tools, which were known long before, already had helical edges; however, the edge was made by grinding. Nonetheless, producing a replaceable carbide insert with a helical edge by means of pressing technology only was a serious problem at that time, and its solution by ISCAR brought about revolutionary changes in the cutting tools industry.
***HELIMILL** – a line of new milling tools was born.*

Endmills with Indexable Inserts

The main functional features of the most popular ISCAR endmills with indexable inserts are summarized in Tables 3 and 4. Table 3 shows the number of teeth of the tools as a function of the tool diameter and its maximal depth of cut; and Table 4 is intended for estimating the rampdown ability of the tools.

Machining cavities is a typical feature of die and mold making. Hence, the ability to drill with subsequent milling is an important attribute of a milling tool. Another significant factor is a tool capability to ramp milling when the tool is simultaneously fed in radial and axial directions.
If the tool moves axially upward, it is ramp up milling (ramping up); and if it moves axially down – ramp down milling. Maximal rampdown angle characterizing tool possibilities in ramping is a valued feature for tool selection. The angle depends on cutting geometry and the nominal diameter of a tool. For tools with high overhang from a toolholder (long and extra long series of the tool overall length) the rampdown angle should be additionally diminished.
Three axes milling by helical interpolation is a widely used method for machining holes especially with a large diameter. A cutter, which travels simultaneously along three coordinate axes of a machine tool, moves planetary about an internal hole diameter and creates the summarized cutting helical movement.
The helix angle is dependent of the maximal rampdown angle of the tool performed interpolation.
The helical tool path maintains smooth entry in the material and uniform loading of the tool.
The particular case of helical interpolation is two axes milling by circular interpolation when only the planetary movement is made.
The depth of a machined hole depends on the maximal allowed depth of cut of the tool.
Chip evacuation and chip re-cutting are factors that affect a milling tool machined cavities by a different technique.

Milling Tools

Table 3 **Quick Selector for 90˚ Indexable Endmills**

	T290 ELN -05	HP E90AN	T490 ELN -08 T490 E90LN -08	H490 E90AX -09	HM90 E90A	T290 ELN -10
ap	5	7.7	8	8	10	10
Tool D	Number of Teeth (Effective)					
8	1					
10	2	1			1	
12	2; 3	2			1	
14	3				1	
16	4; 5	3; 4	2	2	2	
17					2	
18					2	
20		4; 5	2; 3	3	2; 3	2; 3
21					3	
22					3	
25		5; 7	3; 4	4	2; 3; 4	3; 4
28					4	
30					4	
32		6; 8	3; 5	5	3; 4; 5	4; 5
40		8; 10	4; 6		3; 5; 6	6
50					7	
Inserts	T290 LNMT 05	HP ANKT	T490 LN..T 08	H490 ANKX 09	HM90 AP…10 AP…10	T290 LNMT 10

Table 3 **Quick Selector for 90˚ Indexable Endmills (cont.)**

	H490 E90AX -12	T490 ELN -13	HM90 E90AD	T490 ELN -16	H490 E90AX -17
ap	12	12.5	14.3	16	16
Tool D	Number of Teeth (Effective)				
8					
10					
12					
14					
16					
17					
18					
20			1		
21					
22					
25	2		2		
28					
30					
32	3	3	2; 3	2	2
40	4	3; 4	2; 3; 4	3	3
50	5	4; 5	5	4	4
Inserts	H490 AN..X 12	T490 LN..T 13	HM90 AD..15 AD..15	T490 LN..T 16	H490 AN..X 17

Table 4 **Rampdown Angle for 90˚ Indexable Endmills**

	T290 ELN -05	HP E90AN	T490 ELN -08 T490 E90LN -08	H490 E90AX -09	HM90 E90A	T290 ELN -10
ap	**5**	**7.7**	**8**	**8**	**10**	**10**
Tool D	**Rampdown angle, ˚**					
8	2.5					
10	2.3	2.5			5	
12	2	2.7			32	
14	1.5		Ramp	Ramp	7	
16	1	3.2			15	
17			down	down	15/4.5*	
18					7.5	
20		2.4	prohibited	prohibited	7.5	4
21					7.5/2.8*	
22					7.5	
25		2			5	2.2
28					2	
30					2	
32		1.4			3	1.6
40		1			2.7	1.2
50					2.7	
Inserts	T290 LNMT 05	HP ANKT	T490 LN..T 08	H490 ANKX 09	HM90 AP…10 AP…10	T290 LNMT 10

* Lower values for tools with long overall length

Table 4 **Rampdown Angle for 90˚ Indexable Endmills (cont.)**

	H490 E90AX -12	T490 ELN -13	HM90 E90AD	T490 ELN -16	H490 E90AX -17
ap	**12**	**12.5**	**14.3**	**16**	**16**
Tool D	**Rampdown angle, ˚**				
8					
10					
12					
14	Ramp	Ramp		Ramp	
16					
17	down	down		down	
18					
20	prohibited	prohibited	3	prohibited	
21					
22					
25			11.5		
28					
30					
32			5. 3		6.5**
40			4		4.4**
50			5		3.8**
Inserts	H490 AN..X 12	T490 LN..T 13	HM90 AD..15 AD..15	T490 LN..T 16	H490 AN..X 17

** Valid only when H490 ANKX 1706R15T-FF is used.

Die and Mold

General Characteristics of the Latest ISCAR Lines of Indexable Tools for 90° Milling

No mismatch

In machining square shoulders by indexable milling tools, the shoulder height can be more than a tool depth of cut (D.O.C.) that is determined by the length of cutting edge of an insert clamped in the tool. In this case the shoulder machining needs two or more continuous passes. Ensuring true 90° shoulder profile without a detectible border, steps, marks or burrs between the passes – no mismatch – is an essential feature of modern precise indexable milling tools intended for milling square shoulders.
Use of such tools substantially improves productivity by eliminating additional finish cuts and ensures the high-accuracy profile with good surface finish. Perpendicularity of the shoulder wall to its base of no more than 0.02 mm is today a normal requirement to the "no mismatch" milling tools.

HELITANG T490 is a family of milling tools that uses tangentially clamped inserts with four right-hand helical cutting edges. The **T490** inserts are available in 8, 12.5 and 16 mm long cutting edges. The smallest tool diameter is 16 mm, with 2 teeth. The line offers the tools in different configuration: end and face mills, replaceable milling heads for ISCAR **MULTI-MASTER** and **FLEXFIT** systems; and the tools are available with coarse or fine pitch. Most of the tools are provided with holes for internal coolant supply. The tools are intended for milling square shoulders at high rates, with no mismatch and they are capable of plunging as well.
The family features high durability and outstanding tool life, due to the tangentially oriented pocket, the most advanced insert production technology and excellent grade combination.
It is interesting to note that none of ISCAR competitors can offer such a small tangential insert as the **T490 LN**.. 0804PN-R with four cutting edges.

HELIDO H490, a family of tools for 90° milling, is an evolution of the original ISCAR **HELIMILL** line. The H490 AN.. X laydown (radial) insert has 4 right-hand cutting edges. There are 3 standard sizes of the insert with cutting edge lengths 8, 12 and 16 mm.
Insert construction is very thick and strong. It is clamped into a dovetail inclined pocket, which provides a very rigid clamping, and has a wiper that leaves an excellent surface finish. Due to its strong construction, unique chip deflector with positive rake angles and good grade combinations, the family stands out high durability, low cutting forces and long tool life. The tools, which are produced with coolant holes, can machine 90° shoulders with no mismatch, plunge with large stepover and can perform milling slots and faces.
This family combines the most advantageous features of both the **HELIMILL** – helical cutting edge and positive rake angles; and the **MILL2000** – strong construction, most suited for heavy milling applications.

3P ISCAR Premium Productivity Products

Since 2007 this symbol started coming into view on packages of various ISCAR products and on pages of leaflets. The symbol stands for new tools, inserts and toolholders, which are manufactured in accordance with the advanced designed principles and the latest technology, and allow to the customer substantial productivity increase.
In addition to the progressive cutting geometry, the key features of the 3P cutting inserts are the new SUMO TEC carbide grades with their special post-coating treatment.

SUMOMILL T290 is a family of milling tools with coolant holes that uses tangentially clamped inserts with two cutting edges. The **T290** inserts are available in 5 and 10 mm long cutting edges. The inserts are the next evolution of the most popular **HELIMILL** inserts. As a result of their tangential orientation in the pocket, the inserts allow tool design with larger core diameter, providing a much stronger tool construction, which sustains a higher impact load and reduces the risk of fracture to a minimum. When compared to the current **HELIMILL** and **HELIPLUS** tools of similar sizes, the tools of the line feature high long-term strength and remarkable tool life, due to the tangential orientation of the inserts in the pockets and higher tooth density.

The tools runs at higher feed speeds (table feeds), producing excellent surface finish and no mismatch; and are suitable for plunging and rampdown applications. Their increased radial and axial rake angles lead to a major reduction in cutting forces, improve tool stability and prolong life of the cutting edge.

The **T290** unique convex shaped insert enabled the development of small diameter cutting tools of even 8 mm diameter.

Tangential or radial?

The question: what is more effective – tangential or radial clamping, often can lead the user to hesitate when selecting the right cutting tool when there are both milling cutters with laydown (radial) inserts and with inserts clamped tangentially. As in many practical cases, the question has no strictly unambiguous answer. The academic studies of the question are beyond the scope of the guide. Therefore, a brief review of advantages and disadvantages of each clamping principle can be useful for the right choice. In general, the tangential configuration allows increasing feed per tooth because the tangential component of a cutting force acts against an insert with more rational orientation of its cross-section.

The well-designed tangential insert contributes to optimal loading of a clamping screw while the resultant cutting force is transmitted directly to the cutter body. The tangential clamping enables cutter design with larger core diameter than tools with radial inserts. It is much easier to provide an indexable double-sided insert with helical cutting edges if a tangential configuration is applied. And finally, the tangential clamping configuration also offers a higher insert density. Typically, the milling cutters with tangentially clamped inserts run at high feed rates especially when machining cast iron.

However, relative to milling tools with the laydown (radial) inserts, the cutters with the tangential clamping normally have lesser ramping abilities, and their resources for shaping rake face and high positive axial rake, which can significantly reduce cutting forces, are limited. Greater chip gullets in case of the radial inserts go a long way towards better chip evacuation when milling materials such as steel with high metal removal rate, particularly in machining deep cavities.

The latest ISCAR design provides the customers with the milling tools with tangential and radial inserts for which in many cases the disadvantages related to tangential and radial clamping correspondingly were overcome. For example, the H490 line with double-sided radial H490 AN…X inserts, real workhorses, are intended for heavy milling operations with high feed per tooth; and T290 line that is based on tangential T290 LN…T inserts is notable by excellent rampdown performance. In either event the question of using the cutters with tangentially or radially clamping inserts should be solved specifically. The ISCAR application specialists will be glad to advise you the best choice.

Milling Tools

How to Start: Cutting Data

General Principles

Cutting speed Vc and feed per tooth fz, the first key parameters in milling, depend on different factors.

Before everything else, a carbide grade. The harder grade has higher wear resistance and enables higher cutting speed. The tougher grade with its better impact strength is intended for the lesser speed, but allows greater feed per tooth.

Machinability of engineering materials is different, and even the same material can be substantially different by its machinability (for example, milling a tool steel in different conditions: annealed, pre-hardened and hardened). Therefore, a specific force needed for removal of a unit of a chip section, and load acting on an insert differ too. It is evident; the machinability factor should be taken into consideration.

Insert geometry is also important. A sharp cutting edge, which is brittle and can not stand up against a serious load, sets the upper limit to the feed. The T-land, a negative land protecting the cutting edge, conversely, binds the feed below because too small a feed causes in this case, considerably increased cutting force.

Another factor - the milling tool body. A durable design of the body and a reliable method of securing the insert ensure machining under high cutting data.

Further, the application. What is the aim of tool use? In rough milling, when a relatively large volume of material is removed, the feed is high and the cutting speed is moderate. At the same time, finish milling operations performing with small allowances demands maximal speeds and small to medium feeds for high machining accuracy and good surface finish. Different limitations such as large overhang (in milling deep cavities, for example), improper clamping, workpiece with a thin wall and others lead to decreasing the speed, the feed or even both.

Lastly, the machine tool and toolholding. Poor machine conditions and not rigid toolholders create an additional barrier for increased cutting data.

The mentioned arguments are very general; and no doubt everyone who is involved in metal cutting is familiar with them. They are a good illustration of complex dependence or the cutting data on different attributes. How to go from the generalities to the particulars and specify the starting cutting data?

We can take into account the application factor and the machine tool conditions by introducing operation estimation: light, medium and heavy. Then we can prepare the tables with the recommended cutting speed and feed for every milling line and thus provide the user with the cutting data. It is a correct approach; and the cutting data recommendations in full application guides, tool advice software and specific guidelines use it in full.

In the U.S.A., instead of the term “feed per tooth” (“fpt”), “chip load” is often used; “advance per revolution” means “feed per revolution” (“fpr” or “feed”) and “advance per minute” or “feed rate” – “feed speed” (“feed per minute”, “fpm”, or “table feed”).

Starting Feed

Table 5 contains data for estimating starting feed per tooth. Smaller values more suitable for finish operations and greater feeds usually characterize rough milling. In case of high tool overhang and unstable technological system (poor clamping, cutting thin wall workpieces and so on), the feed should be reduced by 20-30%.

Starting Speed

$$Vc = Vo \times Ks \times Kt \quad (1)$$

Where: Vc – starting cutting speed
Vo – basic cutting speed
Ks – stability factor
Kt – tool life factor

a) Basic cutting speed Vo

Table 9 determines the basic cutting speed depending on carbide grade, workpiece material and type of machining. The basic cutting speed relates to a 20 minute tool-life period. In order to define what is known as light duty, moderate duty and heavy duty machining more rigorous, we introduce the following two-step procedure.

Tooth Loading

In shoulder milling, tooth loading is a function of the ratios:

- Cutting depth h to a length of cutting edge ap
- Width of cut b to the nominal diameter of a milling tool D

The tooth loading reflects a fraction of the cutting edge involved in cutting and a cycling path of the tooth in the workpiece material from the tooth entering to the tooth exit.
While remaining in the material too long, the tooth experiences more intensive heat loads that affect the tool life.
The diagram shown in Fig. 1 allows defining the tooth load, but for a quick rough estimate, Table 6 may be enough.

Type of Machining

The tooth load in combination with the feed per tooth defines the type of machining (Table 7).
The feed interval from fz min to fz max relate to a field of the estimated starting feeds as above.
Do not take literally “minimum”, “maximum” and “moderate” feeds: the feeds closed to the lower border of the field relate to fz min, the average values – to fz moderate, and the feeds closed to the upper border – to fz max.

Table 5 **Estimated Starting Feed fz**

ISO Class DIN/ISO 513	Workpiece material				Harder ◄——► Tougher Starting feed fz, mm/tooth, for grades					
	Type	Mat. Group*	σ_T N/mm²	Hard-ness, HB	IC808	IC5100	IC810	DT7150	IC830	IC330
P	Plain carbon steel	1-4	<850	<250	0.1-0.25	0.1-0.25	0.1-0.25		0.1-0.35	0.1-0.4
		5	>850 <1000	>250 <300	0.1-0.25	0.1-0.25	0.1-0.25		0.1-0.3	0.1-0.4
	Alloy steel and tool steel	6, 7	<1000	<300	0.1-0.25	0.1-0.2	0.1-0.25		0.1-0.3	0.1-0.4
		8, 9	>1000 <1200	>300 <350	0.1-0.2	0.1-0.18	0.1-0.2		0.1-0.25	0.1-0.3
		10	<850	<250	0.08-0.18	0.08-0.15	0.08-0.18		0.1-0.25	0.1-0.25
		11	>1100 <1450	>325	0.08-0.15	0.08-0.12	0.08-0.15		0.08-0.2	0.08-0.25
M	Martensitic s.s.	12, 13	<850	<250	0.08-0.12				0.08-0.15	0.08-0.2
K	Grey cast iron	15-16	<1000	<300		0.2-0.35	0.2-0.4	0.2-0.4	0.2-0.3	
	Nodular cast iron	17-18	<1000	<300		0.2-0.35	0.2-0.4	0.2-0.4	0.2-0.3	
H	Hardened steel	38.1	>1480 <1700	HRC 45-49	0.07-0.12				0.07-0.1	
		38.2	>1700 <2000	HRC 50-55	0.06-0.09					
		39	>2000 <2500	HRC 56-63	0.05-0.08					

* ISCAR material group in accordance with VDI 3323 standard

☐ - First choice for grades

For T290 milling tools the table values should be reduced by 30%.

For the face mills with inserts HP ANKX…07 and T490 LN..T…08 the table values should be reduced by 20%.

For milling hardened steel, see the appropriate chapter for further discussion.

Fig. 1. *Tooth load areas*

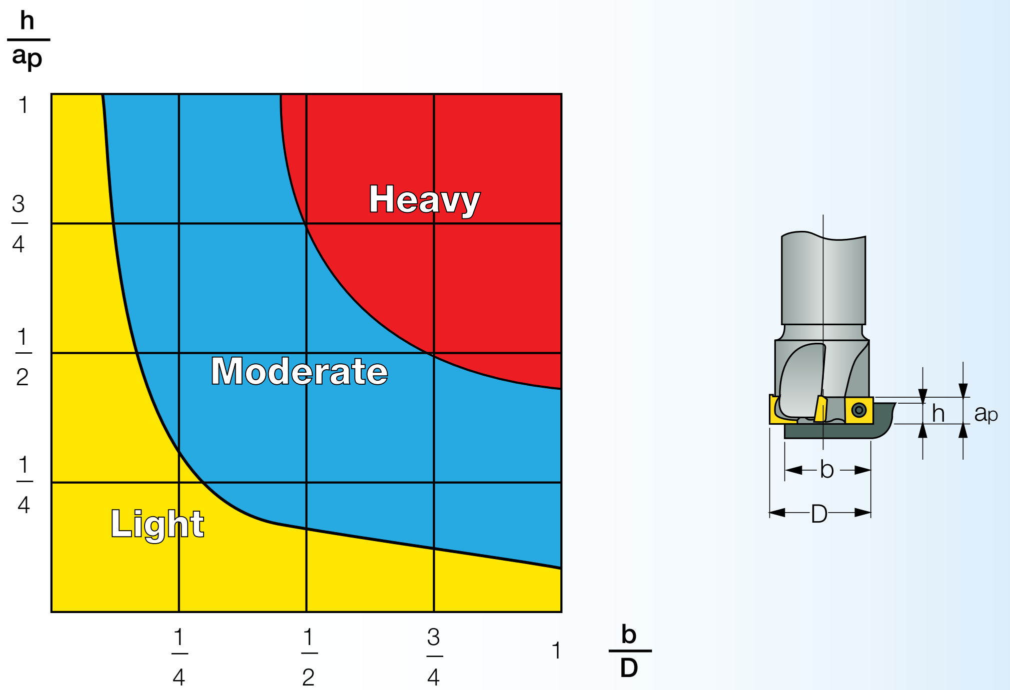


Table 6 **Tooth Load**

b/D	h/ap			
	1/4	1/2	3/4	1
1/4	light	moderate	moderate	moderate
1/2	moderate	moderate	moderate	heavy
3/4	moderate	heavy	heavy	heavy
1	moderate	heavy	heavy	heavy

Table 7 **Type of Machining**

Tooth loading	Type of Machining for Feed per Tooth fz		
	fz min	fz moderate	fz max
Light tooth loading	Light-duty (L)	Light-duty (L)	Medium-duty (M)
Moderate tooth loading	Light-duty (L)	Medium-duty (M)	Heavy-duty (H)
Heavy tooth loading	Medium-duty (M)	Heavy-duty (H)	Heavy-duty (H)

b) Stability Factor Ks

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

- for normal stability Ks = 1,
- for unstable operations (high overhang, poor clamping, milling thin walls, etc.) Ks = 0.7

c) Tool Life Factor Kt

The factor that relies on the relationship cutting speed-tool life is shown in Table 8

Table 8 **Tool life factor Kt**

Tool life, min.	10	20	40	60
Kt	1.15	1	0.85	0.8

Example

The workpiece from AISI P20 steel with hardness HRC 32 is machined by ISCAR indexable endmill cutter H490 E90AX D25-4-C25-09. The cutter carries inserts H490 ANKX 090408PNTR IC830. The application – rough to semi-finish milling of a square shoulder with 4 mm depth and 16 mm width. The workpiece is properly clamped; and the stiffness of the whole technological system (machine tool + fixture) is estimated as sufficient.

The machined material relates to the ninth material group (No.9).
In accordance with Table 5 starting feed fz =0.2 mm/tooth (the value that is close to the upper border).
The length of the cutting edge for the insert above is 8 mm (from the catalog or Table 3).
Hence, h/ap=4/8=0.5 and b/D=16/25≈0.6.
From Table 6 the tooth load is moderate; and from Table 7 the type of machining takes "medium-duty" definition. Ks is accepted as 1 (paragraph b).
Therefore, as per Table 9 starting cutting speed for 20 min. tool life Vc=135 m/min.
For 60 minute tool life, tool life factor Kt=0.8 (Table 8); and the starting cutting speed in this case will be 108 m/min.

Table 9 **Basic Speed Vo for Selected Grades in Relation to Type of Machining***

ISO Class DIN/ISO 513	Workpiece Material				Basic speed Vo, m/min, for grades and type of machining																	
	Type	Material group**	σ_T, N/mm²	Hardness HB	IC808			IC5100			IC810			DT7150			IC830			IC330		
					L	M	H	L	M	H	L	M	H	L	M	H	L	M	H	L	M	H
P	Plain carbon steel	1	<850	<250	300	240	220				260	230	200				200	170	150	185	160	135
P	Plain carbon steel	2-4	<850	<250	280	220	200				240	200	180				180	150	135	170	140	125
P	Plain carbon steel	5	>850 <1000	>250 <300	240	200	180				215	190	170				150	135	120	135	120	115
P	Alloy steel and tool steel	6, 7	<1000	<300	230	200	170				200	180	160				170	140	125	150	135	120
P	Alloy steel and tool steel	8, 9	>1000 <1200	>300 <350	215	185	165				180	150	125				150	135	120	140	125	115
P	Alloy steel and tool steel	10	<850	<250	210	190	170				165	135	110				140	125	115	130	120	110
P	Alloy steel and tool steel	11	>1100 <1450	>325	165	135	115				150	125	105				135	120	115	125	110	100
M	Martensitic s.s.	12, 13	<850	<250	200	170	140										170	140	125	150	130	120
K	Grey cast iron	15-16	<1000	<300	260	220	200	300	250	220	300	250	220	300	250	220	260	220				
K	Nodular cast iron	17-18	<1000	<300	240	200	180	250	220	200	250	220	200	250	220	200	200	185				
H	Hardened steel***	38.1	>1480 <1700	HRC 45-49	120	100	80										100	80	70			
H	Hardened steel***	38.2	>1700 <2000	HRC 50-55	75	55																
H	Hardened steel***	39	>2000 <2500	HRC 56-63	65	45																

* For 20 min. tool life.
** ISCAR material group in accordance with VDI 3323 standard
*** Milling hardened steel see under the appropriate chapter for further discussion.
☐ - First choice for grades

Solid Carbide Endmills

ISCAR offers the die and mold makers a rich line of solid carbide endmills with 90° lead angle for machining square shoulders. These tools with nominal diameters from 0.4 mm to 25 mm and varied in form are intended for machining all types of materials used in the die and mold industry such as tool and alloy steels, martensitic stainless steel, cast iron, etc. The tools differ in cutting geometry, helix angle, number of flutes and length series (short to extra long) and perform all kinds of milling: rough, semi-finish and finish operations.

ISCAR catalogs and leaflets contain detailed guidelines for using the solid carbide endmills in shoulder milling. In general, recommended practice says that if the endmills with two flutes, which have largest chip gullet characteristics, are intended mostly for rough shoulder machining, milling slots and plunging; the multi-flute mills are usually used in finish applications with high requirements of accuracy and surface quality.

Commonly, a tool choice and a cutting data depend on application requirements and workpiece material. However, when speaking about feed limitations, it should be emphasized that the main factors are not only tooth strength and tool rigidity, but also the ability of chip handling that is defined by a chip gullet (a form of flute and its depth). Hence, due to the mentioned factors the feed is limited by the nominal diameter and the number of flutes of a solid carbide endmill.

Endless cutting tool manufacturers from small shops to world-known companies produce solid carbide endmills of the same sizes that often seem like copies of each other. However, in spite of a formal resemblance, occasionally simply amazing, there is a great difference in performance and tool life of the mills. The reason lies in carbide grades, grinding technology and of course, unique features of cutting geometry.

Dry or wet

Dry machining (or using air as a coolant) is preferable for solid carbide endmills. In machining steels and hard steels (ISO classes P and H correspondingly) by mills of IC900 and IC903 carbide grades, a wet coolant is not recommended. If, however, an application requires wet cooling (machining austenitic stainless steel, for instance), grade IC300 should be a first choice.

There are several carbide grades for ISCAR solid endmills.

The majority of the solid mills is produced from IC900 – a tough submicron substrate with PVD TiAlN coating, which has wide-spectrum rough to finish applications at medium to high cutting speeds in milling carbon, alloy, tool and stainless steel. Also, the grade is suitable for milling hardened steel with hardness to HRC 55.

IC903 with an ultra-fine grain substrate with 12% cobalt content and PVD TiAlN coating can be recommended for milling hardened steel, especially if its hardness is HRC 56-63 and even more. The grade should not be used for heavy-duty machining.

IC300 is a tough submicron TiCN PVD coated grade that is suitable for machining steel and stainless steel workpieces, specifically under unfavorable conditions, at low to medium cutting speeds.

Uncoated fine-grain grade IC08 is aimed mainly at milling nonferrous materials.

Table 10 **Grade Selector for Solid Carbide Mills**

ISO class		IC900	IC903	IC300
P		■		□:
M		□		■
K		■		
H	Up to HRC 55	□	■	
	Above HRC 55		■	

■ - First choice
□ - Optional

Within the rich and diversified family of ISCAR solid carbide square endmills (with 90˚ lead angle), the following three lines have exceptional geometry.

The **FINISHRED** endmills feature 4 flutes with a 45˚ helix, two serrated teeth and two continuous teeth, combine two geometries: rough (serrated teeth with chip splitting effect) and finish (continuous teeth) and therefore sometimes are called “Two in One”. They enable running at rough machining parameters, resulting in semi-finish or even finish surface quality. Such a single tool (“One”) can replace the rough and finish endmills (“Two”), dramatically reducing cycle time and power consumption, and increasing productivity. The unique tool design reduces vibrations at heavy-duty applications; and the chip mixture from long and split short chips is evacuated more easily, which is a good solution for machining cavities of dies and molds.

The **CHATTERFREE** endmills have 4 or 5 flutes and unequal tooth spacing. Due to the uneven arrangement of the teeth these tools feature excellent dampening ability. They provide an effective solution for low power machine tools with ISO40 or BT40 adaptations, which are popular in small works involved in die and mold making. The mills are able machine full slots with 2xD depth.

The **FINISHRED VARIABLE PITCH** combines all the remarkable features of the two lines above. Combining the lines together delivers a powerful hybrid solid carbide mill with extraordinary performance. The user gets a “Three in One” rather than just a “Two in One” version of the **FINISHRED**.

Regrinding solid carbide endmills

The progress in technology of grinding machine tools led to impressive achievements in regrinding (resharpening) solid carbide endmills, allowing accurate restoration of cutting geometries of worn-out tools. Reground mills still have shorter tool life due uncoated areas or problematic recoating. However, the main reason of the efficiency losses is a reduction of a tool diameter as a result of regrinding the tool relief surfaces. Regrinding the relief surfaces causes decreasing rake angles and flute depth. Therefore, the tool cuts harder; its chip handling properties become worse. On average, every 1% decrease of the tool diameter results in a decline of the tool performance by 2%-3%, and from the definite reduction value the tool performance drops dramatically. In order to avoid negative sides of regrinding it is very important to follow ISCAR’s instructions regarding this operation.

Cutting Data

Maximal Depth for Milling Full Slot

In full slot milling, the maximal depth depends not only on strength and stiffness of a mill but on chip handling properties. In many cases milling deep slots demands considerable reduction of feed per tooth in order to ensure proper chip evacuation. Using **FINISHRED** and **FINISHRED VARIABLE PITCH** can substantially improve the mentioned difficulty.
The maximal depth should not exceed the values shown in Tables 11 to 13. The values relate to slot milling in steel; and they should be doubled for workpieces from cast iron.

Table 11 **Maximal Depth of Slot Ap max, mm, for S.C. Mills of Standard Line**

D, mm	to 4	4-5	6-8	10-25
Ap max	0.3 D	0.4 D	0.4 D / D*	0.5 D / D*

* with fz reduced by 50%

Table 12 **Maximal Depth of Slot Ap max, mm, for FINISHRED Mills**

D, mm	to 4	4-5	6-8	10-25
Ap max	0.4 D	0.6 D	0.7 D / 1.2 D*	0.9 D / 1.5 D*

* with fz reduced by 30%

Table 13 **Maximal Depth of Slot Ap max, mm, for CHATTERFREE and FINISHRED Variable Pitch Mills**

D, mm	to 4	4-5	6-8	10-25
Ap max	0.9 D	D	1.2 D	2 D*

* for D≥16 mm fz should be reduced by 20%

Milling Square Shoulder: Dimensional Limitations

In rough to finish shoulder milling, depth of cut ap and width of cut ae can be estimated in accordance with Table 14.

Table 14 **Shoulder Milling: Shoulder Size**

ae	ap max	
	D ≤ 16 mm	D > 16 mm
< 0.3 D	2 D	1.8 D
(0.3…0.5) D	2 D	1.8 D
0.5 D < ae < 0.75 D	1.25 D	0.8 D
≥ 0.75 D	Ap max*	

☐ - Recommended operational mode

* Ap max as it specified in Tables 11-13

Starting Feeds and Speeds

The following tables specify estimated values for feed per tooth and cutting speeds referring to **FINISHRED**, **CHATTERFREE** and **FINISHRED** Variable Pitch endmills.

The tables relate to rough and semi-finish milling. The recommendations regarding finish operations are discussed on the next pages separately.

In case of unfavorable conditions (poor clamping, milling thin walls, high overhang), the table values should be reduced by 20-30%.

Table 15 **FINISHRED: Starting Feed fz, mm/tooth, for Mill Diameters D**

ISO Class DIN/ISO 513	D, mm												
	Mat. Group*	1	2	3	4	5	6	8	10	12	16	20	25
P	1-4	0.008	0.020	0.030	0.040	0.055	0.065	0.080	0.085	0.092	0.138	0.145	0.155
	5	0.008	0.020	0.028	0.038	0.055	0.065	0.077	0.082	0.090	0.130	0.137	0.148
	6, 7	0.008	0.020	0.028	0.038	0.055	0.065	0.077	0.082	0.090	0.130	0.137	0.142
	8, 9	0.008	0.019	0.028	0.038	0.050	0.060	0.072	0.077	0.085	0.130	0.137	0.142
	10	0.008	0.017	0.025	0.036	0.048	0.058	0.070	0.072	0.082	0.125	0.130	0.137
	11	0.008	0.012	0.022	0.032	0.045	0.055	0.065	0.065	0.077	0.110	0.120	0.132
M	12, 13	0.008	0.015	0.028	0.038	0.048	0.058	0.070	0.077	0.082	0.125	0.130	0.137
K	15-16	0.009	0.022	0.032	0.043	0.060	0.070	0.083	0.088	0.095	0.142	0.150	0.163
	17-18	0.009	0.022	0.032	0.043	0.060	0.070	0.083	0.088	0.095	0.142	0.150	0.163
H	38.1**				0.022	0.028	0.032	0.038	0.040	0.045	0.055	0.060	0.065
	38.2												
	39												

* ISCAR material group in accordance with VDI 3323 standard

** HRC 45-49

Slot drills

Endmills that can cut straight down are also called “slot drills”. They have at least one center cutting tooth, and their primary use is for milling key slots (“slot drilling”). Normally, the slot drills are two-flute mills, but often they have three and sometimes even four flutes.

Table 16 **CHATTERFREE and FINISHRED Variable Pitch Starting Feed fz, mm/tooth, for Mill Diameters D**

ISO Class DIN/ISO 513	D, mm										
	Mat. Group*	3	4	5	6	8	10	12	16	20	25
P	1-4	0.022	0.032	0.045	0.055	0.058	0.060	0.065	0.078	0.088	0.110
	5	0.022	0.032	0.040	0.050	0.055	0.058	0.060	0.065	0.077	0.100
	6, 7	0.020	0.027	0.032	0.042	0.045	0.050	0.055	0.060	0.072	0.088
	8, 9	0.019	0.027	0.032	0.036	0.038	0.042	0.050	0.055	0.065	0.075
	10	0.016	0.022	0.027	0.030	0.032	0.038	0.045	0.050	0.060	0.067
	11	0.012	0.016	0.022	0.027	0.030	0.035	0.038	0.045	0.055	0.060
M	12, 13	0.016	0.022	0.027	0.030	0.032	0.038	0.045	0.050	0.060	0.067
K	15-16	0.022	0.032	0.045	0.055	0.055	0.062	0.072	0.082	0.100	0.130
	17-18	0.019	0.027	0.042	0.052	0.052	0.060	0.068	0.078	0.090	0.110
H	38.1**			0.016	0.020	0.022	0.025	0.027	0.032	0.035	0.045
	38.2										
	39										

* ISCAR material group in accordance with VDI 3323 standard
** HRC 45-49

Table 17 **Starting Speed Vc, m/min (rough to semi-finish milling)**

ISO Class DIN/ISO 513	ISCAR Mat. Group*	Slot Milling	Shoulder Milling
P	1	145	180
	2-4	115	150
	5	100	125
	6	105	130
	7-9	85	120
	10	85	115
	11	70	100
M	12, 13	80	110
K	15-16	130	160
	17-18	125	150
H	38.1**	50	70

* ISCAR material group in accordance with VDI 3323 standard
** HRC 45-49

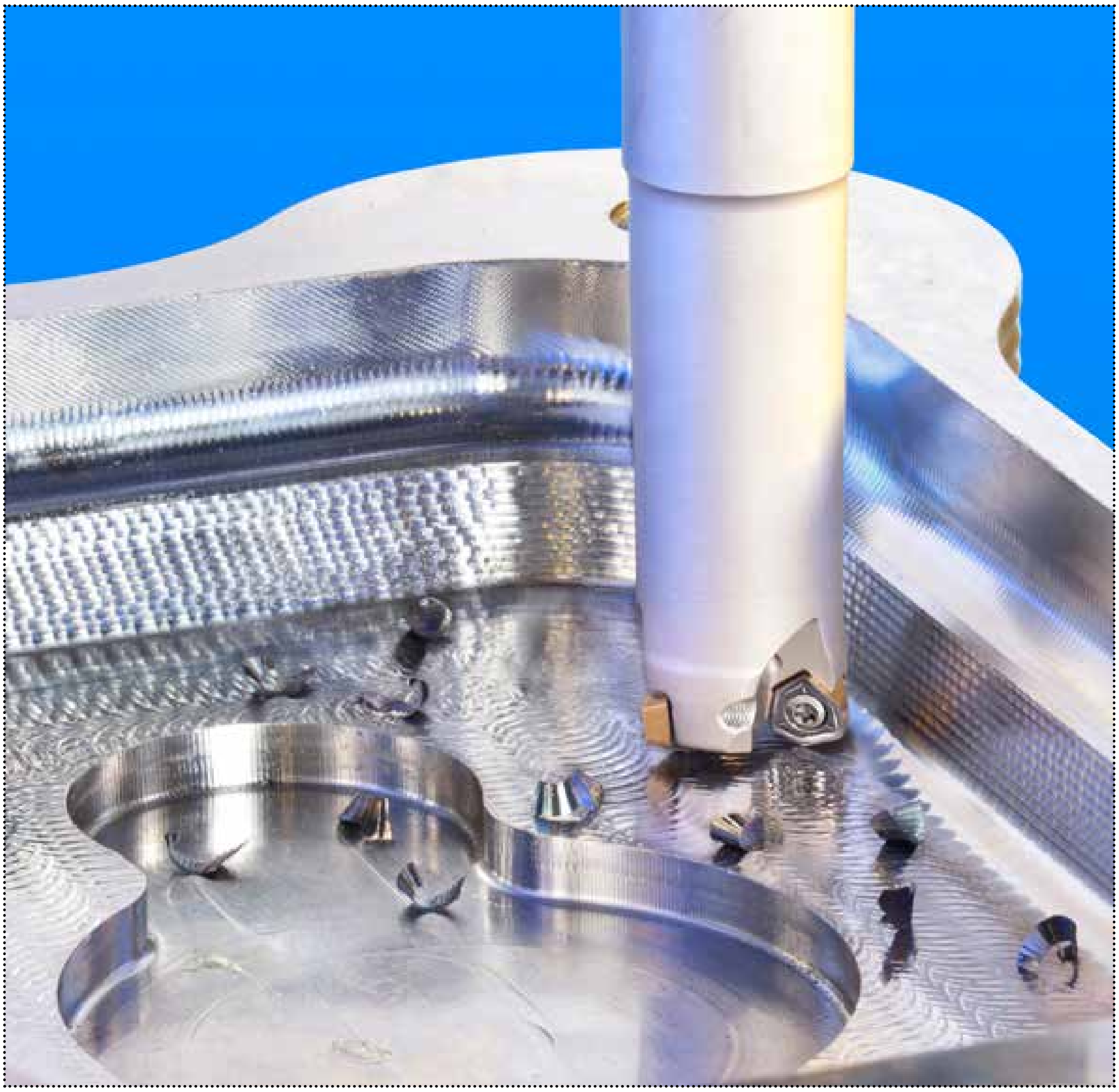
Ramp down milling and solid carbide endmills

We have already underlined that in die and mold making, where machining cavities and pockets are so widely used, the ramp down ability is a very important feature of a milling tool. It stands to reason that the slot drills have no limitations regarding a ramping angle, but the ramp down characteristics of other endmills shall be thoroughly examined before planning and CNC programming the corresponding milling operations (helical interpolation, for example) – refer to ISCAR catalogs or product guidelines for appropriate data.

Table 18 **Starting Speed Vc, m/min (finish milling)**

ISO Class DIN/ISO 513	ISCAR Mat. Group*	Vc, m/min
P	1	280
	2-4	200
	5	170
	6	190
	7-9	170
	10	165
	11	120
M	12, 13	150
K	15-16	220
	17-18	200
H	38.1	100
	38.2	90
	39	60

* ISCAR material group in accordance with VDI 3323 standard
** HRC 45-49



Finishing

Solid endmills as integral, monolith tools ensure high dimensional and form accuracy (tolerance limits for a tool diameter, runout of teeth relative to the tool shank, etc.). Therefore, they fully meet the requirements for finish milling (or finishing) of die and mold parts. The typical features of finishing operations are high accuracy and surface quality of machined surface, and small allowances (to 5% of a mill diameter and 0.1-0.2 mm for hardened steels). In finishing, the cutting speed is high and the feed per tooth is low relative to rough and semi-finish operations. The tool strength allows cutting with feeds greater than in Table 15 and 16, but due to insufficient surface finish it is recommended to start cutting under the table values and then try to increase until the surface roughness is enough.

Short or Extra-Long Reach?

Solid carbide endmills of the same type and nominal diameter vary in flute lengths and overall lengths. The mills of short length ensure highest strength and rigidity whereas the extra long reach mills are designed for deep cavities and high shoulders. As a rule, a series of standard endmills comprise short, medium, long reach and extra long reach tools.

Example

It is required to mill a square shoulder of 3 mm width (ae) and 5 mm height (ap) in a mold block from AISI/SAE 4340 steel with hardness HRC 34. The required roughness is Ra 2.5, the accuracy requirements to straightness and flatness of the shoulder walls in accordance with ISO 2768-m (medium). The available tool: solid carbide endmill EC-E4L 08-18/26W08CF63.

The machined material relates to the eighth material group (No.8).
The nominal diameter of the above mill is 8 mm (D), its cut length 18 mm (Ap).
The mill relates to the **CHATTERFREE** solid mill line.
The specified requirements to the milled surface are not high; ae/D = 3/8, ap<Ap, ap<2D (5<16); hence the shoulder can be machined by one pass (Table 14).
From Table 16 starting feed fz =0.038 mm/tooth and from Table 17 starting cutting speed Vc=120 m/min.

Feed per tooth or depth of cut?

In milling, metal removal rate, the litmus test of productivity, depends both on feed per tooth and on depth of cut. The question: "What is more effective for effective productivity control – varying the feed or the depth within the acceptable limits?" has no unambiguous answer.
But in general, under the same metal removal rate, increasing the feed coupled with reduced depth of cut is more favorable than the opposite combination (the lesser feed with the deeper cut) because it normally results in greater tool life.
By the way, high feed milling (HFM), one of the progressive rough milling techniques that are also taken into consideration in this guide, rests in particular on this principle.

Table 19 **General Characteristics of Solid Carbide Endmills**

No. of flutes	Strength	Rigidity	Chip handling	Roughing	Finishing	Slot milling	Plunge milling
2	*	*	****	****	*	****	****
3	**	**	***	***	**	***	**
4	***	***	**	**	***	*	
5 and more	****	****	*		****		

For more detailed information regarding 90° solid carbide endmills, refer to ISCAR catalogs, guides and technical leaflets.

MULTI-MASTER Endmill Heads

General Notes

MULTI-MASTER is a family of tools with shanks and interchangeable cutting heads for a variety of machining applications: milling, countersinking, spot and center drilling, and slitting.
The **MULTI-MASTER** design approach bases upon a thread system of the unique profile, centering by a short precise taper and a face contact. A **MULTI-MASTER** head has a cutting part and a back connection with the external thread and the taper, which screws into a shank with the corresponding internal thread and the taper until final securing when the back face of the head cutting part will contact the face of the shank.
This principle of coupling ensures strength and rigid clamping of a wide range of the interchangeable heads. The **MULTI-MASTER** tools meet the requirements of high accuracy because the head geometry is finished by precise grinding and the connection guarantees high concentricity within very close limits. In addition, the tools are simple-to-operate, because the heads are quickly replaced by easy rotation of an applied key. Moreover, they answer to strict requirements of repeatability, and thus, replacement of the heads does not require additional adjustment.

The **MULTI-MASTER** family features a large variety of heads, shanks and extensions. The basic concept is, when a shank can carry heads of different shapes and accuracy, this allows dramatic increase of tool versatility and will diminish needs for special tools. A huge stock of tools is not necessary. Resharpening of tools is no longer needed, because a worn-out cutting head is simply replaced. The family renders a possibility of numerous tools by an unlimited combination of the heads and the shanks and therefore, excellently answers the demands of die and mold making and reduces procurement cost.

No setup time advantages

*Repeatability of an assembled mechanical system with interchangeable elements means that a key parameter of the system remains in agreed limits in case of replacing an interchangeable element of the same type. For the standard **MULTI-MASTER** tools, repeatability in tool length is about 0.04 mm for the milling heads of normal accuracy and about 0.02 mm for the precise milling heads.*
That is why there is no need for additional adjustment in tool length after replacing a head; and the head can be replaced when a shank remains clamped in a machine tool spindle without new presetting. No setup time for replacement considerably cuts cycle time and is a good source for increasing productivity.

90° Endmill Cutting Heads

There are two kinds of the relatively small-diameter (8-25 mm) **MULTI-MASTER** heads for square shoulder milling.

The first, which is designated **MM EC**…, has exactly the same cutting geometry (number of flutes, helix angle, etc.) as the solid carbide endmills.
The only difference is a smaller cutting length: normally, it does not exceed a head diameter.
It goes without saying that every type of solid carbide endmill has also been produced as a **MULTI-MASTER** head.
Naturally, the cutting data for **MM EC**… heads is the same as for the 90° solid carbide endmills considered in the previous pages.

The second, "economy" type, designated **MM HC**… features only two flutes and a lesser helix angle. Being pressed and sintered to shape and size, the cutting geometry of the heads of this type is merely finished by grinding. Due to the high-impact structure of a pressed tooth the heads run at feeds per tooth that are significantly greater than in case of solid carbide endmills or **MM EC**… heads, so despite only two teeth, the feed speed is the same as for multi-flute mills or heads. The mentioned strength property allows even a slight increase of cutting speed relative to the solid carbide tools/heads for the same tool life period.
This property makes **MM HC**… heads to be an attractive economical solution especially in rough milling and slot drilling.
In milling slot or square shoulder with the use of **MM HC**… heads, the natural geometrical limitation is cutting length Ap of a head. At any rate it is not recommended to exceed 0.8Ap for depth of cut while milling shoulders with widths more than a half of the head diameter or milling slots.
Tables 20 and 21 show data for estimated starting feeds and speeds for **MM HC**… heads.

Indexable solid carbide tools

The endmills that are assembled from the ***MULTI-MASTER*** *heads and shanks open new doors to saving money and improving productivity.* ***MULTI-MASTER*** *modular tools, which are neither solid carbide nor indexable tools in a popular sense, lay in the intermediate field between them.*
Having a replaceable solid carbide cutting part they relate to a new type of cutting tools: indexable solid tools, unlikely as it may seem with the combination of the words "indexable" and "solid".

Table 20 **MM HC… Heads: Starting Feed fz, mm/tooth**

ISO Class DIN/ISO 513	fz,mm/tooth, for D, mm				
	Mat. Group*	8	10	12	16
P	1-4	0.11	0.13	0.13	0.15
	5	0.1	0.12	0.13	0.13
	6, 7	0.09	0.1	0.1	0.12
	8, 9	0.08	0.09	0.1	0.12
	10	0.07	0.08	0.09	0.1
	11	0.06	0.07	0.08	0.09
M	12, 13	0.07	0.08	0.09	0.1
K	15-16	0.1	0.13	0.14	0.16
	17-18	0.09	0.12	0.13	0.15
H	38.1**	0.04	0.05	0.06	0.06

* ISCAR material group in accordance with VDI 3323 standard
** HRC 45-49

Milling Tools

Die and Mold

Milling Tools

MULTI-MASTER shanks

*The **MULTI-MASTER** shanks are produced from various materials:*
steel (for general applications), tungsten carbide (has high stiffness) and heavy metal (an alloy with great tungsten percentage with small alloying additions of other metals; the alloy features excellent vibration damping properties but not recommended for heavy-duty applications due to its limited impact fatigue strength).
The shanks differ in configuration: without neck and with straight or tapered neck.
The taper angle for standard shanks varies from 5° on side to 1°. Of course, overall lengths and neck lengths vary also.
Combining the above shank characteristics provides the die maker with the tool that is exactly needed in his specific operation, whether it be roughing or finishing, machining deep cavities or shoulders, milling under poor clamping or with high overhang.
*Modularity converts the **MULTI-MASTER** tools into a powerful means for choosing the right tool.*

Table 21 **MM HC... Heads: Starting Speed Vc, m/min (rough to semi-finish milling)**

ISO Class DIN/ISO 513	ISCAR Mat. Group*	Slot Milling	Shoulder Milling
P	1	160	190
	2-4	125	160
	5	110	135
	6	115	140
	7-9	95	130
	10	95	125
	11	80	110
M	12, 13	90	120
K	15-16	145	175
	17-18	135	165
H	38.1**	55	75

* ISCAR material group in accordance with VDI 3323 standard
** HRC 45-49

In case of unfavorable conditions (poor clamping, milling thin walls, high overhang) the values in Tables 20 and 21 should be reduced by 20-30%.

Generally, the **MULTI-MASTER** endmill heads are produced from grade IC908 that has tough submicron substrate and PVD TiAlN coating. This universal grade is suitable for wide-range milling applications, including interrupted cut and unfavorable conditions, with medium to high cutting speeds; and feature excellent notch wear and built-up edge resistance.
Additionally, some heads intended for milling hardened steel are manufactured from grade IC903. The previous section is devoted to solid carbide endmills and contains a brief description.

Example
Rough milling a square shoulder of 4 mm width (ae) and 6 mm height (ap) is need during machining a bushing of a die set. The bushing material – steel AISI S1 with hardness HB 190. A planner specified for milling a **MULTI-MASTER** tool comprising short steel shank MM S-A-L070-W20-T10 and economy head MM HC160C16R0.4-2T10 908. The bushing is properly clamped into the clamping fixture of the machine tool.

The machined material relates to the sixth material group (No.6).
The nominal diameter of the above head is 16 mm (D), its cutting length 15 mm (Ap).

Under these tool dimensions and machining requirements one pass is enough for milling the shoulder (ae/D = 6/16=3/8, ap<Ap); so the shoulder can be machined by one pass (p. 31).
From Table 20, starting feed fz =0.12 mm/tooth and from Table 21, starting cutting speed Vc=140 m/min.

*For more detail information regarding **MULTI-MASTER** endmill cutting heads, shanks and extensions, refer to ISCAR catalogs, guides and technical leaflets.*

90˚ Extended Flute Mill Cutters

Cutting blades of extended flute mill cutters consist of sets of indexable inserts. The inserts, which are placed gradually and with mutual offset of one another, produce continuous helical cutting blades and engage the material during machining and results in a smooth cut. As against an ordinary indexable mill with length of cut that is limited by the cutting edge of the insert mounted on the mill, the cutting length of the extended flute cutter is significantly larger – it is "extended" due to the whole set of the insert. Correspondingly, a chip gullet of the ordinary mill transforms in a full flute for proper chip handling.

For fully effective extended flute cutters the number of face teeth is equal to the number of flutes; and for half effective cutters with staggered order of the inserts, the number of the face teeth is twice as small as the flute number. Understandably, the fully effective cutters run twice as fast as the half effective ones under the same cutting speed and feed per tooth.

The extended flute cutters can be with integral body or modular tools assembled from different sections. In accordance to sizes and adaptation the cutters are produced with shanks or like shell mills.

The extended flute cutters perform heavy roughing operations such as heavy-duty milling deep slots, grooves and shoulders. They are used also as edging mill.

ISCAR offers the broad-range line of the fully effective extended flute mill cutters. Although this type of the milling tools is not the most common in the die and mold making process, within the scope of the reference guide we consider briefly the assignment of cutting data for the extended flute cutters with the latest inserts. The detailed method of defining speeds and feeds in this case demands taking into account different combinations of the main machining parameters, which can adversely affect performance and tool life. Therefore, for initial estimation it is quite sufficient to use the data of Tables 22-24 that relate to the extended flute cutters carrying the inserts of SUMO TEC carbide grades.

ap in the tables is depth of cut, ae means width of cut correspondingly, and D – nominal cutter diameter.
The speeds for milling full slot take values of milling shoulder with ap/D>0.5; and if in this case ap/D>1.25, the values should be reduced by 20%.

Cutting porcupine

In the technical literature the extended flute mill cutters are called also long-edge, porcupine-type or porcupine cutters and even, as it is sometimes called in shoptalk, "porkies".

Table 22 **Starting Speed Vc, m/min (average data for extended flute cutters)**

ISO Class DIN/ISO 513	ISCAR Mat. Group*	ae/D for grades											
		IC808			IC810			IC830			IC330		
		<0.3	0.3…0.5	>0.5	<0.3	0.3…0.5	>0.5	<0.3	0.3…0.5	>0.5	<0.3	0.3…0.5	>0.5
P	1	225	200	170	200	180	160	160	140	125	150	130	115
	2-4	200	175	160	180	160	150	140	125	115	125	115	110
	5	170	155	140	155	140	130	120	110	100	110	100	100
	6	180	165	130	165	150	120	130	115	95	120	110	95
	7	170	150	130	155	135	120	125	110	95	115	105	85
	8	170	140	125	155	130	115	120	105	90	115	100	85
	9	140	125	120	130	115	110	105	90	85	100	85	80
	10	140	120	115	130	110	105	105	90	85	100	85	80
	11	120	105	95	110	100	90	85	80	75	80	75	70
M	12, 13	125	110	100				110	100	90	110	100	90
K	15-16	175	160	135	200	170	150	170	150	130			
	17-18	165	140	125	170	150	135	160	135	120			
H	38.1**	70											

☐ – First choice for grades

* ISCAR material group in accordance with VDI 3323 standard

** HRC 45-49

Table 23 **Basic Feed fo, mm/tooth, for Extended Flute Cutters**

ISO Class DIN/ISO 513	ISCAR Material Group*	fo, mm/tooth							
		T290 LNK…		T490 LNK	(LNM)…	T490 SM…		H490 SM…	
		12-16	20	20-32	40-50	40-50	63	63	80-100
P	1	0.13	0.14	0.16	0.2	0.2	0.22	0.22	0.25
	2-4	0.11	0.12	0.14	0.17	0.17	0.18	0.18	0.19
	5	0.09	0.1	0.12	0.15	0.15	0.16	0.16	0.17
	6	0.09	0.1	0.12	0.15	0.15	0.16	0.16	0.17
	7	0.09	0.1	0.12	0.15	0.15	0.16	0.16	0.17
	8	0.09	0.1	0.12	0.15	0.15	0.16	0.16	0.17
	9	0.08	0.09	0.1	0.12	0.12	0.13	0.13	0.14
	10	0.08	0.09	0.1	0.12	0.12	0.13	0.13	0.14
	11	0.06	0.07	0.08	0.1	0.1	0.1	0.1	0.12
M	12, 13	0.12	0.1	0.12	0.15	0.15	0.16	0.16	0.16
K	15-16	0.14	0.12	0.14	0.17	0.17	0.19	0.19	0.2
	17-18	0.12	0.1	0.12	0.15	0.15	0.16	0.16	0.17
H	38.1**	0.05	0.06	0.07	0.08	0.08	0.09	0.09	0.1

* ISCAR material group in accordance with VDI 3323 standard

** HRC 45-49

Table 24 **Loading Factor K_L for Extended Flute Cutters**

ap/D	ae/D				
	1/10	1/4*	1/2	3/4	1
1/20	1.2	1	0.9	0.8	0.75
1/4	1.2	1	0.9	0.8	0.7
1/2	1.2	1	0.8	0.7	0.65
3/4	1	0.8	0.7	0.6	0.55
1	0.9	0.7	0.6	0.55	0.5
>1.25	0.7	0.6	0.5	0.45	0.4

* For T290 LNK… cutters ae/D shall not exceed 1/4

Starting feed per tooth fz for the extended flute cutters can be estimated in accordance with the following equation:

$$fz = fo \times k_L \qquad (2)$$

Where: fo – basic feed (Table 23)

k_L –loading factor (Table 24) that takes into consideration tool loading.

Example

Extended flute shell mill cutter H490 SM D100-64-5-40-17C carrying inserts H490 ANKX 170608PNTR IC830 performs rough milling of a deep shoulder of a large-size die structural component made from AISI/SAE 4130 alloy steel, which has hardness HB 270. The shoulder dimensions are as follows: 60 mm depth and 40 mm width.
The machine tool power is sufficient, the clamping conditions are good.

The machined material relates to the seventh material group (No.7).
The nominal diameter of the cutter is 100 mm (D), its maximal cutting length 61.5 mm (Ap) - the catalog data. The inserts' carbide grade – IC830.

ae/D = 40/100=0.4; ap/D = 60/100=0.6.
Hence, from Table 22, starting cutting speed Vc can be estimated as 110 m/min; and from Table 23, basic feed fo=0.17 mm/tooth. Table 24 specifies loading factor k_L as 0.75 (approximation).
Thus the starting feed fz = 0.17x0.75≈0.13 mm/tooth.

Important Note

Heavy-duty rough milling with the extended flute cutters, especially of shell mill type, often demands high machine power. Check power consumption and compare it with the main drive specification of your machine tool!

Tangential or radial? (2)

Further to our discussion regarding tangential and radial (laydown) insert clamping in the context of the extended flute cutters, it can be noted that under otherwise equal conditions the extended flute cutters with radial clamping have certain advantages in chip handling and can show better results while milling full deep slots. The cutters with tangential clamping, however, provide an advantage when machining deep shoulders due to their more rigid body structure.
Of course, these remarks should not be considered as absolute guidelines for choosing a cutter or as a final solution of the question of the header, but is a good illustration of a reason for different principles of tool configurations.

2 Milling Plane Surfaces

The first choice for milling plane surfaces is face mills. The face mills are intended for machining surfaces that are parallel to the mill face. Obviously, the endmill examined in the previous section, also belongs to such general definition. Moreover, the face mill and likewise the endmills machine square shoulders. But the definition of mill types relates to main distinctive features and underlines them in specification.

Due to main application the face mills have additional design features in comparison with the endmills. The cutting diameter of a face mill is considerably larger than its maximal depth of cut; and the ratio between the mentioned key dimensions far exceeds the corresponding ratio in endmills. Generally, the face mills are designed as shell mills with a stepped bore for mounting directly upon an adapter, an arbor or a machine tool spindle; and have a key slot on its non-cutting face for torque transfer. The endmills, on the contrary, belong to shank-type tools for which mounting and activation is performed by shank.

One of the main design characteristics of a milling tool is the position of the cutting edge of the tool with respect to machine surfaces. The position defines the cutting edge angle (entering angle). So as to meet various shape requirements of machined parts, cutting tool manufacturers produce the face mills with different cutting edge angles. The most common types of the face mills have the following cutting edge angles:

- 90° (0° lead angle correspondingly)
- 75° (15° lead)
- 60° (30° lead)
- 45° (45° lead)

Classically, if there is no limitation regarding the shape of machined surface, the face mill with the smallest cutting edge angle is recommended. The point is that the maximal chip thickness, which characterizes the mill loading, is equal to full feed per tooth for 90° mills and, for cutting edge angles differing 90°, to the product of the feed to sine of the cutting edge angle. Therefore, the smaller angles allow higher feed per tooth under the same mill loading (maximal chip thickness) – a good source for increasing productivity*. See tables 25, 26.

Table 25 **Chip Thickness for the Same Feed per Tooth**

Cutting edge angle	90°	75°	60°	45°
Chip thickness	100%	97%	87%	71%

Table 26 **Feed Increasing for the Same Chip Thickness**

Cutting edge angle	90°	75°	60°	45°
Feed per tooth	100%	103%	115%	141%

* This approach resides in high-feed machining (HFM).

Cutting edge angle or approach angle?

There a well-defined strict specification for the angles' defined cutting geometry of the milling tools. However, national standard sometimes defines the same parameters in a different manner that causes misunderstandings even in technical literature and between experienced metal cutting specialists.
The angle between the plane of the main cutting plane (the plane that the edge machines) and the machined plane surface (which the mill machined) is called the tool cutting edge angle or the cutting edge angle.
Some technical sources call it also the entering angle, the cut-entering angle, the entry angle or the entrance angle. Its value is often indicated before a general type or header of milling tool families. For example: "90° milling tool" means that the mill has a 90° cutting edge angle.

In the U.S.A. and U.K., it is common practice to specify the angle that is complimentary to the cutting edge angle: so-called the lead angle (USA) and the approach angle (UK) – the angle formed by the main cutting edge and a line parallel to the axis of the cutter rotation.

Being the complimentary angles, the cutting edge angle and the lead angle produce together 90°; and they are equal only in case of 45° mills. For our example above the lead angle is 0° ("zero lead").

As it is not always clear what angle is specified, please pay attention to prevent confusion.

In addition, the smaller cutting edge angles lead possibly render a smoother engagment into the material and exit during milling operations. The radial component of a cutting force is reduced, and the radial pressure, correspondingly, too. That is very important for materials that have a tendency of break-out edges (cast iron, for instance). Heat dissipation is better; insert life is longer.

However, many applications demand 90° shoulder shapes. Decreasing the cutting edge angle reduces the radial component of the cutting force but leads to increasing its axial component*, which can cause deflected machined surfaces, especially if a workpiece has thin walls. Moreover, other insert dimensions being more or less equal, 90° mills have greater maximal depth of cut – an important feature of the mill.

It stands to reason that milling operations need a different type of mill; and only correctly understanding operation requirements coupled with the main properties of the different mill types will bring the proper choice of mill.

ISCAR has a rich line of face mills with various cutting edge angles for broad spectrum applications. Among the latest 90° face mills are the tools of the **HELITANG T490**, **SUMOMILL T290** and **HELIDO H490** families. The main features of these tools have already been discussed in brief in the section of the endmills in this guide. As before, **HELI2000 HM90** family with the classically designed 2 cutting edged insert is in great request.

The 45° face mills grew in the newest **HELIDO S845** family with an innovative solution that enables using either square or octagonal double-sided inserts on the same tool. The mills of the family offer an effective solution of inserts with 8 or 16 cutting edges. If the square right-hand inserts feature durable structure, positive geometry of the rake face and 8 cutting edges; the octagonal high-strength inserts have 16 cutting edges, either right-hand or left-hand and are very economical per edge inserts.

Summary tables 27 and 28 show main general data of some advanced ISCAR 90° and 45° face mills.

* This approach resides in high-feed machining (HFM).

Table 27 **Quick Selector for Certain 90° Indexable Face Mills**

	HP F90AN	T490 FLN -08	H490 E90AX -09	HM90 F90AP	H490 E90AX -12	T490 FLN -13	HM90 F90A	T290 FLN -15	T490 FLN -16	H490 E90AX -17	HP F90AT -19
ap	**7.7**	**8**	**8**	**10**	**12**	**12.5**	**14.3**	**15**	**16**	**16**	**16**
Tool D	**Number of Teeth (Effective)**										
32	6; 8	3; 5	5	3; 5							
40	8; 10	4; 6	6	5;6	4	4; 5	3; 4	4	3	3	
50	9; 12	5; 7	7	6; 7	3; 5	5; 6	3; 5	5	3; 4	3; 4	3; 4
63	12; 16	6; 9	9	7; 9	4; 6	6; 8	4; 6	6	4; 6	4; 6	4; 5
80				8; 11	5; 7	7; 10	5; 7	7	5; 7	5; 7	4; 6
100				9; 13	6; 9	8; 13	6; 8		5; 8	5; 8	5; 7
125				10; 16		9;17	7; 9		7; 10	7; 10	
160							8; 10			8; 12	
200							9; 12			10	
250							10			12	
315							12				
Inserts	HP ANKT 07	T490 LN..T 08	H490 ANKX 09	HM90 AP…10	H490 AN..X 12	T490 LN..T 13	HM90 AD..15	T290 LN..T 15	T490 LN..T 16	H490 AN..X 17	HP90 AD.. 1906

Wiper insert

Normally, the ordinary inserts for face milling are provided today with a wiper flat for improving quality of a machined plane surface. This is a small secondary cutting edge, which in spite of the definition "flat", sometimes has a complex geometry but can be considered to be parallel to the machined surface. For this, although the wiper flat improves the surface roughness, it is not always enough, particularly for the face mills with a relatively large nominal diameter.

The situation becomes much better if in addition to the ordinary inserts, a face mill carries a specially designed insert (or more rarely two for large-sized tools), for which the wiper flat is significantly larger than for the ordinary one. This wiper insert is mounted in the same pocket but it protrudes 0.05…0.07 mm relative to the ordinary inserts towards to the machined plane. Due to the wide wiper flat the wiper insert applies additional force to the workpiece that needs special emphasis for milling brittle materials.

Table 28 **Quick Selector for HELIDO 45° Indexable Face Mills**

	SOF45 8/16	SOF45 8/16	S845 F45SX
ap	**3.5**	**6**	**7.1**
Tool D	**Number of Teeth (Effective)**		
40	4	4	4
50	4; 6	4; 6	4; 5
63	6; 8	6; 8	5; 7
80	7; 10	7; 10	6; 9
100	8; 12	8; 12	7; 11
125	10; 16	10; 16	8; 14
160			10; 18
200			12
250			15
315			19
Inserts	ON…U 0505	S845 SN…U 1305	S845 SXMU

Table 28 **Quick Selector for HELIDO 45° Indexable Face Mills (cont.)**

	HP F90AN	**T490 FLN -08**	**H490 E90AX -09**	**HM90 F90AP**	**H490 E90AX -12**	**T490 FLN -13**	**HM90 F90A**	**T290 FLN -15**	**T490 FLN -16**	**H490 E90AX -17**	**HP F90AT -19**
ap	**7.7**	**8**	**8**	**10**	**12**	**12.5**	**14.3**	**15**	**16**	**16**	**16**
Tool D	**Number of Teeth (Effective)**										
32	6; 8	3; 5	5	3; 5							
40	8; 10	4; 6	6	5;6	4	4; 5	3; 4	4	3	3	
50	9; 12	5; 7	7	6; 7	3; 5	5; 6	3; 5	5	3; 4	3; 4	3; 4
63	12; 16	6; 9	9	7; 9	4; 6	6; 8	4; 6	6	4; 6	4; 6	4; 5
80				8; 11	5; 7	7; 10	5; 7	7	5; 7	5; 7	4; 6
100				9; 13	6; 9	8; 13	6; 8		5; 8	5; 8	5; 7
125				10; 16		9;17	7; 9		7; 10	7; 10	
160							8; 10			8; 12	
200							9; 12			10	
250							10			12	
315							12				
Inserts	HP ANKT 07	T490 LN..T 08	H490 ANKX 09	HM90 AP…10	H490 AN..X 12	T490 LN..T 13	HM90 AD..15	T290 LN..T 15	T490 LN..T 16	H490 AN..X 17	HP90 AD.. 1906

Pitch of milling tool

The pitch is the distance between two nearest-neighbor teeth of a milling tool measured between the same points of the cutting edges of the teeth. The pitch shows density of tooth of a milling tool in accordance to which the tools differ from the tools with coarse, fine and extra fine pitch.

Choosing the proper right tooth density depends on two main things. On the one hand, at least one tooth should always cut material. On the other hand, the space of a chip gullet should be enough for chip handling. Hence, the machined material and type of machining are the principal factors for choosing. General-duty coarse pitch with maximum chip gullet space and a few teeth are the first choice for rough and finish milling of steel. Also, in many cases the coarse pitch is a way to reduce vibration.
Fine pitch, which provides less space for the chip gullet but more teeth, allows increased feed speed. It is recommended for milling cast iron and for applications requiring limited feed per tooth.
Extra fine pitch generally is suitable for milling steel with shallow depth of cut, milling cast iron and high-efficiency machining with considerable feed speed.

Sometimes, in order to overcome vibrations during operations, the teeth are produced with unequal spacing, which is to say, the tool has differential pitch.

In technical literature parallel with the coarse-fine-extra fine pitch, grading also exists: coarse-medium (or regular)-fine, normal-close-extra close and other definitions. In addition, tools with extra-fine pitch can be called high-density cutters.

Cutting diameter and width of cut

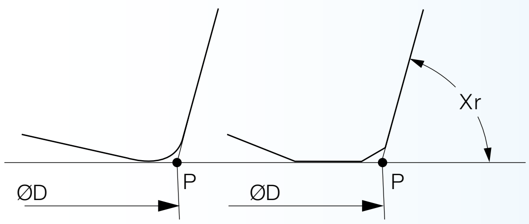
Normally, peak efficiency in face milling can be achieved if a width of cut will be 60-80% of a mill cutting diameter. Of course, the real situation often dictates other proportions even when milling a full slot, but the above ratio is always preferable.

The meaning of the cutting diameter for the face milling tool with indexable inserts needs clarification. In accordance with ISO standard 6462 it shall be taken from point P as defined in Fig. 2 (diameter D). Point P is the theoretical point produced by intersection of the main cutting edge and the plane machined by the mill. The cutting diameter is one of the main dimensions of the mill, and it often is called the nominal diameter.

For 90° face mills the cutting diameter is consistent along the cutting edge and does not change for different depths of cut (ap).

At the same time, if the cutting edge angle (χr) differs from 90°, the real cutting diameter will be D+2×ap×tan χr. One would think that the right calculation should take this point into consideration. But the fact that in the face mills the maximal depth of cut of a tool far less cutting diameter D, allows neglecting the above correction; and it is conventional to specify the cutting speed, the spindle speed, etc. with respect to cutting diameter D as defined in ISO 6462.*

Fig. 2. *Cutting diameter D and cutting edge angle (in accordance with ISO 6462)*



* The guide also uses this specification.

Initial Cutting Data

Basically, the way of defining the initial cutting data is the same as for the endmills with indexable inserts, already examined.

a) Starting feed per tooth fz

$$fz = fz_0 \times k\chi \qquad (3)$$

Where: fz_0 – the basic starting feed
$k\chi$ – the factor of the cutting edge angle

The basic starting feed can be chosen in accordance with Table 5, that specifies the feed limits for the more popular ISCAR carbide grades. Optionally, as the case may be, the satisfactory estimation of the basic feed is achieved with the use of Table 29, which contains the average values for the face mills.

Table 29 **Basic Starting Feed fzo for Face Mills**

ISO Class DIN/ISO 513	Workpiece material		Basic feed fzo, mm/tooth, for grades					
	Type	Mat. Group*	IC808	IC5100	IC810	DT7150	IC830	IC330
P	Plain carbon steel	1-4	0.2	0.2	0.2		0.22	0.25
		5	0.18	0.18	0.18		0.2	0.22
	Alloy steel and tool steel	6, 7	0.15	0.15	0.18		0.2	0.22
		8, 9	0.12	0.12	0.15		0.18	0.2
		10	0.12	0.12	0.12		0.15	0.15
		11	0.1	0.1	0.12		0.12	0.15
M	Martensitic s.s.	12, 13	0.1				0.12	0.15
K	Grey cast iron	15-16		0.22	0.25	0.25	0.22	
	Nodular cast iron	17-18		0.2	0.22	0.22	0.2	
H	Hardened steel	38.1	0.08				0.08	
		38.2	0.07					
		39	0.05					

* ISCAR material group in accordance with VDI 3323 standard
☐ – First choice for grades
For T290 face mills the table values should be reduced by 30%.
For the face mills with inserts HP ANKX…07 and T490 LN..T…08 the table values should be reduced by 20%.
For milling hardened steel, see the appropriate chapter for further discussion

The factor of the cutting edge angle $k\chi$ (Table 30) reflects possible increasing of feed for the same chip thickness due to position of the main cutting edge relative to the machined plane surface (compare with Table 26).

Table 30 **Factor of the Cutting Edge Angle** $k\chi$

Cutting edge angle	90°	75°	60°	45°
$k\chi$	1	1	1.1	1.4

b) Starting cutting speed Vc

Exactly as for the indexable endmills, calculation of starting cutting speed Vc for the face mills rests on equation (1):

$$Vc = Vo \times Ks \times Kt$$

Where: ***Vc*** – starting cutting speed
Vo – basic cutting speed
Ks – stability factor (1 if stability is enough and 0.7 for unstable operations)
Kt – tool life factor (Table 8)

Table 9 specifies basic cutting speed Vo as dictated by the workpiece material, the carbide grade and the type of machining, whilst defining the latter can be done through Tables 6 and 7. If desired, upon condition that the starting feed was calculated from equation (3) and Table 29, the type of machining for a face mill more suitable for the operation (when the width of cut is 60-80% of the mill diameter) or for milling a full slot (100% diameter engagement), can be taken from Table 31.

Table 31 **Type of Machining: Face Milling***

h/ap	b/D			
	60%	**70%**	**80%**	**100%****
1/8	Light-duty (L)	Light-duty (L)	Light-duty (L)	Medium-duty (M)
1/4	Light-duty (L)	Medium-duty (M)	Medium-duty (M)	Medium-duty (M)
1/2	Medium-duty (M)	Medium-duty (M)	Heavy-duty (H)	Heavy-duty (H)
3/4	Heavy-duty (H)	Heavy-duty (H)	Heavy-duty (H)	Heavy-duty (H)

* For starting feed fz in accordance with equation (3)
** Milling full slot
Be aware: milling with a width of cut that is 60%-80% of a mill diameter ensures peak efficiency in face milling

Shoptalk: facing and shouldering

In the art of metal cutting, face milling is often called simply facing, slot milling – slotting, and shoulder milling, correspondingly - shouldering.

Example

A face of a plate 90 mm × 406 mm from nodular cast iron DIN GGG 50 (ASTM 65-45-12) with hardness HB 180 is machined by face mill cutter SOF 8/16-D125-16-40R that carries inserts ONMU 050505-TN IC810. The depth of cut is 3 mm.
Power of the machine tool main drive is sufficient; the plate is secure mounted in the clamping device of the machine tool.

The machined material relates to the seventeenth material group (No.17).
The nominal diameter of the cutter is 125 mm (D), its maximal cutting length 3.5 mm (ap) - the catalog data. The inserts' carbide grade – IC810.
The cutting edge angle of the face mill cutter is 44° (it can be rounded off to 45°).

h/ap = 3/3.5=0.86; b/D = 90/125=0.72.
So, basic starting feed fzo=0.22 mm/tooth (Table 29), factor of the cutting edge angle k_χ=1.4 (Table 30) and starting feed fz = 0.22x1.4≈0.3 mm/tooth.
The type of machining is heavy-duty (Table 31), and starting cutting speed Vc= 200 m/min (Table 9).

Example

Cutter H490 F90AX D063-6-27-17 with inserts H490 ANKX 170608PNTR IC330 performs face milling of a large-scale block by several passes, for which the depth of cut is 6 mm and the width of cut is 50 mm. The block material is carbon steel AISI/SAE 1030, HB 200…220. The operation stability can be estimated as sufficient.

The machined material relates to the second material group (No. 2).
The nominal diameter of the cutter is 63 mm (D), its maximal cutting length 16 mm (ap) - the catalog data. The inserts' carbide grade – IC330.
The cutting edge angle of the face mill cutter is 90°.

h/ap = 6/16=0.38; b/D = 50/63=0.8.
Then basic starting feed fzo=0.2 mm/tooth (Table 29), factor of the cutting edge angle k_χ=1 (Table 30) and starting feed fz = 0.2 mm/tooth.
The type of machining according to Table 31 lays between medium- to heavy-duty, for which Table 9 recommends the starting speeds 140 and 125 m/min correspondingly. As it stands we can accept a more or less average value between these limits as the starting cutting speed, thus Vc = 130 m/min.

Clamping screws with adjustable protrusion nozzle for shell-type face m ills

Most conventionally designed shell-type face mills are equipped with coolant holes for the spindle-through coolant supply. The holes are usually located in the chip gullets and are directed towards the cutting edges. This design ensures coolant supply at the cutting zone, but in many cases does not facilitate chip evacuation, as the coolant flow tends to push the chips back towards the inserts.
Several cutting tool manufacturers produce clamping screws with internal ports.
The screws secure the face mills on the adapter and ensure flow through the axis of a tool to its periphery and thus facilitate improved chip evacuation. However, as shell mills have central countersink with different depths, effective application of such screws is limited.
The ISCAR clamping screws with movable nozzle eliminate the above disadvantage.
The protrusion of the nozzle of the screw can be easily adjusted according to countersink depth, insert size or any other application requirements. The nozzle screw position can be secured by a locking-nut.
The adjustable protrusion nozzle effectively directs the coolant supply to the cutting zone, thus improving chip evacuation substantially.

3 Milling Contoured Surfaces (profiling)

The outstanding characteristics of the die and mold industry show through a large proportion of contoured surfaces, especially in the production of plastic molds and die-casting dies. Therefore, while speaking about the specific tools for die and mold, one often means the tools, and first of all milling cutters, for machining the very same contoured surfaces. These kinds of tools represent the majority of metal cutting tools required by die and mold makers.

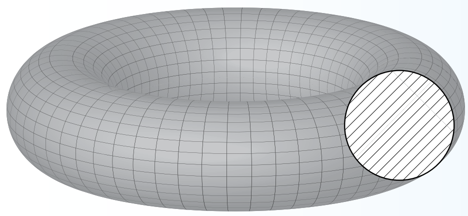
Contoured surfaces

In the technical literature the contoured surfaces are also called formed surfaces, profiled surfaces, 3-D shaped surfaces, and etc.

3.1. Milling tools with toroidal cutting profile

A milling tool with toroidal cutting profile being rotated around the tool axis produces a torus – a geometrical form, which is generated by revolving a circle about an axis that does not touch the circle and is coplanar with it – a ring-shaped bagel (Fig. 3).

Fig. 3.

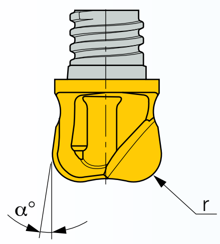


Structurally, this profile is realized in the following designs:

- Milling tools with indexable round inserts
- Solid carbide toroidal milling tools
- Interchangeable solid carbide toroidal milling heads

The solid carbide tools and heads are more suitable for small nominal diameters. Due to some limitations they often look like a bull-nose or back-draft mills (Fig. 4)

Fig. 4.



Milling Tools

3.1.1. Milling tools with indexable round inserts

By all accounts, the milling tools with round (or as it also called button) inserts are the most popular roughing and semi-finishing mills in machining dies and molds, especially complex male-female main parts such as cavities and projections. In roughing, when the primary target is the highest metal removal rate, the round inserts have two significant advantages that contribute greatly to their successful use. Whether or not, both of the advantages relate to the round shape of the inserts.

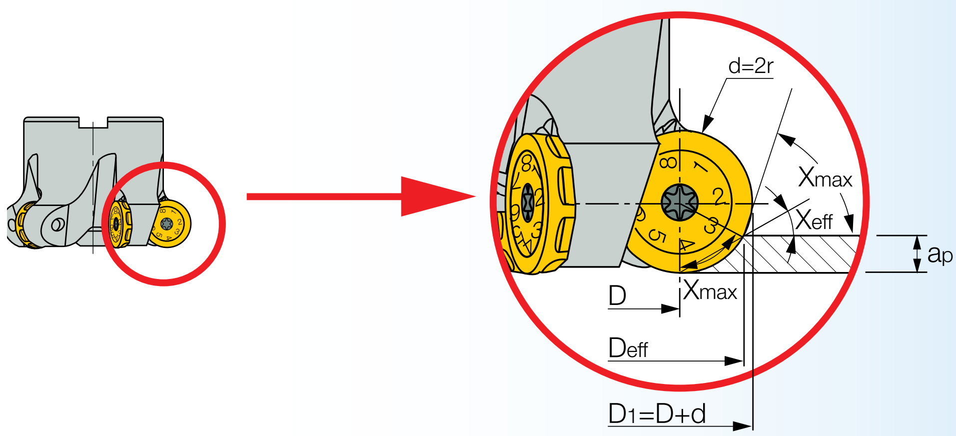
Firstly, strength of the cutting edge. If the cutting corner is the Achilles heel for an insert of any polygonal shape, the round insert without a cutting corner is much stronger.
The higher strength allows more heavy tool load or in other words, stronger cutting data: more feed and speed that cause the improved metal removal rate.

All around … a round!

A tool intended for milling countered surfaces should have a cutting geometry that not only contacts the surface but also ensures correct generation of a needed shape in every direction. Generally speaking, for rough milling any cutting geometry can be suitable; but in finish operations, only the cutting edge, which lays on a spherical portion, meets the above requirements. Therefore the milling tools with complex cutting form: toroidal or ball nose, are very popular in the die and mold industry. One way or another, but look about – a round!

Secondly, a varying cutting edge angle. The cutting edge angle, which is consistant in case of the polygonal insert, varies along the cutting edge of the round insert from zero to the value measured at the highest point of the cutting edge that is involved in cutting. The latter defines the maximal cutting edge angle (χmax, Fig. 5). When the tool runs at its maximum depth of cut that is equal to the insert radius, the maximal cutting edge angle will be 90°.

Fig. 5.



Milling Tools

Die and Mold

The maximal cutting edge angle can be found from the following equation:

$$\cos \chi_{max} = 1 - ap/r \quad (4)$$

Where **ap** – a depth of cut
r – the radius of a round insert

Due to variation of the cutting edge angle along the cutting edge area involved in milling, the average or effective cutting edge angle χeff often used in calculations.

$$\chi_{eff} = \chi_{max}/2 \quad (5)$$

In case of side milling when the face of a tool with round inserts does not cut, the effective cutting edge angle (Fig. 6) is defined as:

$$\chi_{eff} = [\arccos (1-ap_2/r)+\arccos (1-ap_1/r)]/2 \quad (5a)$$

The effective cutting edge angle is one of the important factors of cutting geometry of the tools with round inserts. Similar to the tools with polygonal inserts it determines the relationship between the radial and the axial components of a cutting force. The effective cutting edge angle depends on a depth of cut and therefore, it changes, between a close to zero value for very shallow depths of cut and 45˚ when the depth of cut is equal to the radius of a round insert. Because a change of the depth of cut causes a change of the effective cutting edge angle and hence, of relationship between the cutting force components, this phenomenon allows for an effective way of cutting force monitoring. The growth of the depth causes increasing the radial and reducing the axial components, and vice-versa. Evidently, in very shallow cuts the radial component will be negligible and the cutting force acting on the main cutting plane is directed towards the tool axis.

3P(2): Patent-Protected Products

The absolute majority of new 3P products relates to state-of-the-art development and meets the requirements of up-to-the minute technology. Therefore, these 3P tools and inserts are the products protected by patent or at least are patent-pending products.

The considerable radial component leads to the tool deflection and vibrations, while the excessive axial component adversely affects machining accuracy and can cause problems with workholding. Therefore, the correct choice of the depth of cut allows the die and mold maker to define the optimal loading conditions for machining a specific part depending on the part shape, machine tool characteristics, clamping fixture, etc.

Table 32 **Cutting Edge Angles as Function of Ratio ap/r***

ap/r	1/16	1/8	3/8	1/2	5/8	3/4	7/8	1
Xmax	20˚	29˚	52˚	60˚	68˚	76˚	83˚	90˚
Xeff	10˚	14.5˚	26˚	30˚	34˚	38˚	41.5˚	45˚

* ap is an axial depth of cut, r - the radius of a round insert

Decreasing the cutting edge angle produces more thin chips and thus allows an effective way for higher productivity. Indeed, the feed per tooth used for programming a tool route relates to the maximal possible depth of cut that is equal to the radius of a round insert. In this case a tool with round inserts is more or less similar to a milling tool with 45˚ cutting edge angle. But if the tool cuts under the maximal depth, the chip is thinner (Fig. 7); and the programmed feed per tooth should be increased correspondingly in order to produce the chips of required thickness. That is why chip thinning demands even higher feeds and thus makes improving productivity possible.

Fig. 6.

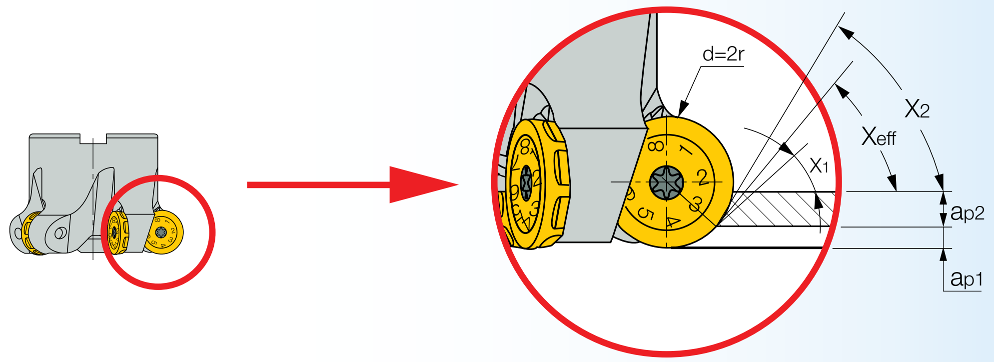
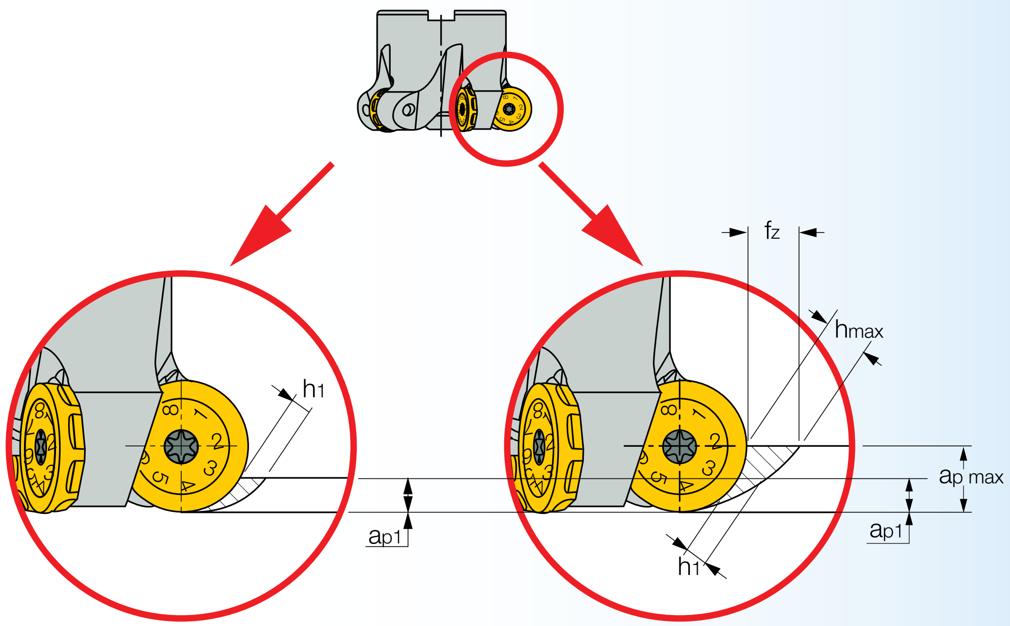


Fig. 7.



Die and Mold

Insert size

The size of a round insert is its diameter. For example, "insert size 12" means "a 12 mm diameter round insert".

Although milling at the full possible depth (the radius of a round insert) leads to maximum engagement between a tool and a workpiece, considerable increase of the radial component of the cutting force, extraneous vibrations, and as a result, excessive wear, - the shape of a machined contoured surface sometimes demands coming into cutting contact exactly at this area. On the other hand, maximum effect is at the highest if the depth of cut ensures that the maximal cutting edge angle 25˚…45˚, that is approximately 0.1…0.3 of the insert radius (or in other words, 0.05…0.15 of the insert diameter). For that reason the ISCAR regular round inserts have 8 indexed working positions for 45˚ or 90˚ indexing. The rake face of some of them is indicated in the corresponding numbers: 1, 2, 3, … for better usability. The specially shaped side faces on the relief surfaces or the bottom protrusions of the inserts keep an insert from drifting during cutting.

High overhang dramatically reduces vibration stability of milling cutters and the mills with round inserts are no exception. As a rule, milling with the overhang already twice as large the mill diameter requires decreasing cutting data for stable machining. The same situation is observed while milling near thin walls and in poor workholding: both of these cases also result in reduced cutting data. **ISCAR MILLSHRED** line offers a good solution for the problem. In addition, it provides the users with another operating benefit of efficient milling at the full possible depth.

The main principle of the **MILLSHRED** insert is a combination of the round cutting profile with the chip splitting effect. It is achieved by the serrated cutting edge of the insert. The insert has 4 indexed working positions for 90˚ indexing; and the serrations on the cutting edge were designed so they overlap, providing a fully effective tool configuration. The MILLSHRED milling tools can carry either the round inserts with serrated cutting edge (designated as RCMT…) or regular round inserts (RCC… MO).

Serrate around

The serrated cutting edge that splits chips into small pieces significantly upgrades the round insert. First of all, reducing the cutting force (and especially its radial component) accrued from the chip splitting, substantially improves the dynamic behavior and less bending of a tool, that ensures stable cutting with normal feeds if the tool overhang is high. Therefore the overhang range can be considerably extended. Normally the upper border of the range comes close to the ten tool diameters. In many cases such a simple and cheap solution renders unnecessary use of complex and expensive toolholding devices with vibration-dampening properties.
It is very important to note that the serrated cutting edge provides a means of depth equal to the insert radius.
Also, the lower the cutting force the lower the machining power.
Secondly, the small chip pieces simplify chip evacuation, specifically in deep cavities of dies and molds.
And thirdly, the small pieces have fewer tendencies to be re-cut and that greatly improves rough milling of the deep cavities and increases tool life.

The selection guideline and general recommendations regarding a more suitable cutting geometry for MILLSHRED plain and serrated inserts depends on machining parameters and the type of workpiece material (listed in Tables 33 and 34).

Table 33 **Selection Guidelines for MILLSHRED Round Inserts**

Insert	RCC...MO	RCMT...
Cutting Edge Type	plain	serrated
Depth of cut to 0.15×d*		
Depth of cut above 0.15×d		
Tool overhang to 2.5×D**		
Tool overhang above 2.5×D		
Milling near thin walls		
Poor workholding		

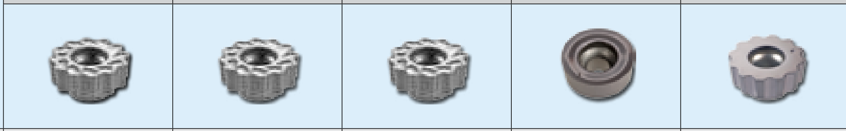
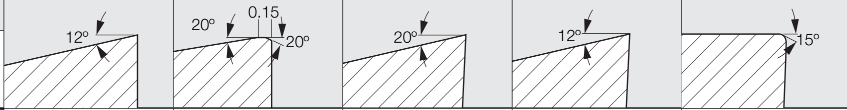
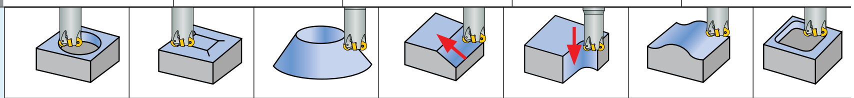
☐ - Recommended choice
* d is the diameter of a round insert
** D is the nominal diameter of a tool

Fig. 8. *A plain round insert mounted for better surface finish*

Round wiper: a plain round insert improves surface finish

Due to the ability to carry both types of round insert, with plain and serrated cutting edge, the ***MILLSHRED*** *cutter allows for combining into one tool assembly. However, so promising an alternate mounting plain and serrated inserts together in one tool (a sort of "FINISHRED") generally does not lead to essential advantages. Nevertheless, if in a tool with the serrated inserts we change a serrated insert on the plain round one, the surface finish of a machined workpiece will be improved. Here the regular plain insert looks like a wiper insert. Although the MILLSHRED tools are intended mostly for rough milling, they are also used for semi-finish operations and the above way of improvement is a simple but effective means. The plain round insert should be mounted in the pocket that is marked by a small round recess (so-called "watermark", Fig. 8).*

Table 34 **Selection of MILLSHRED Inserts for Machining Die and Mold Materials**

Type of Milling			Roughing			Roughing and Semi-Finishing	
Insert Designation			RCMT...FW	RCMT...FW-T20	RCMT...FW-F20	RCCT...MO	RCCW..MO
							
Mat. group	Geometry		Positive rake face	Positive rake face with T-land	Positive rake face sharp edge	Positive rake face	Flat top face with a t-land
	Workpiece		 				
1-5	P	Plain carbon steel	√√√	√	√	√√√	√√
6-9	P	Low-alloy steel	√√√	√	√	√	√√√
10-11	P	High-alloy steel and tool steel	√√√	√	√	√	√√√
12-14	M	Stainless steel		√√√ (1)	√√√ (2)	√√√	
15-20	K	Cast iron	√√				√√√
38.1	H	Hardened steel (HRC 45 max)	√				√
Application							

Application: Helical interpolation | Shouldering | Contour Milling | Ramping Down | Plunging | Profiling | Pocket Milling

(1) Recommended for martensitic stainless steel
(2) Recommended for austenitic stainless steel
√ Last recommended
√√ Second option
√√√ Most recommended

Mainly, the **MILLSHRED** inserts are produced from grades IC908, which has already been mentioned in the previous sections, and IC928 – a PVD TiAlN coated tough grade that is especially recommended for heavy-duty operations. Additionally, the inserts intended for machining cast iron are made from PVD AlTiN coated carbide grade IC910.

Starting Cutting Data

a) Starting feed per tooth fz

i. MILLSHRED plain round inserts

Generally, the starting cutting feed per tooth that should be use for CNC programming can be found from the following equation:

$$fz = fz_0 \times KTH \times Ks \qquad (6)$$

Where: **fz0** – the basic starting feed
KTH – the chip thinning factor
Ks – the stability factor

The basic starting feed (Table 35) is a recommended feed for milling with depth of cut that is equal to the radius of a round insert. It corresponds to a maximal chip thickness, which is planned to be carried out during machining.

Table 35 **MILLSHRED Plain Round Inserts. Basic Starting Feed fzo, mm/tooth**

ISO Class (DIN/ISO 513)	Material Group*	Basic starting feed fzo, mm/tooth, for inserts					
		RC..W...MO			RC..T...MO		
		Ø12	Ø16	Ø20	Ø12	Ø16	Ø20
P	1-4	0.21	0.28	0.34	0.14	0.19	0.23
	5-9	0.21	0.28	0.34	0.14	0.19	0.23
	10-11	0.15	0.2	0.24	0.1	0.13	0.16
M	12 ,13	0.18	0.24	0.29	0.12	0.16	0.2
K	15-16	0.21	0.28	0.34			
	17-18	0.18	0.24	0.29			
H	38.1	0.12	0.15	0.19			

☐ - recommended choice

* ISCAR material group in accordance with VDI 3323 standard

The chip thinning factor, as the function of a depth of cut and hence of a cutting edge angle, reflects the necessity for increasing the basic starting feed; while CNC programming for obtaining the planned maximal chip thickness at a time when machining is performed with a depth of cut less than the radius of the insert.

$$KTH = 1/\sin(0.75 \times \chi max) \qquad (7)$$

or, that is the same,

$$KTH = 1/\sin(1.5 \times \chi eff) \qquad (7a)$$

The stability factor that takes into account the effect of cutting stability is taken to be 1 for normal conditions and to 0.7 if the estimated cutting stability is insufficient (milling thin walls, high overhang, poor toolholding, non-rigid workholding, etc.).

Example

The face of a plate from AISI H13 tool steel annealed to HB 170…190 is machined by shell mill cutter FRW D068A080-07-27-12 carrying inserts RCCW 1206MO IC908 with 1.5 mm depth of cut. The plate is properly clamped into a workholding device and the cutter has light overhang. What is the feed per tooth necessary for CNC programming of the operation?

From Table 35 the basing starting feed for the material group No.10, related to the specified steel, is 0.15 mm. Using equation (4), $\chi max=41^{\circ}$; and equation (7), KTH=1.95.
Hence, fz = 0.15×1.95×1=0.29 (mm/tooth); and the programmed feed should be 0.29 mm/tooth.

Alternatively, the feed can be determined by applying Tables 36 and 37 following.

The tables contain feeds per tooth as a function of the diameter of an insert and a depth of cut. The data in the tables relates to machining a group of materials that is chosen as basic; and for the other material groups the table values should be multiplied by corresponding coefficients.

Table 36, for instance, reflects milling workpiece material groups 5-9 (low alloy steel and quenched plain carbon steel) and 15-16 (grey cast iron) with the use of inserts RC..W…MO. For machining groups 10-11 (high alloy and tool steels), the table values should be multiplied by 0.7, for machining groups 17-18 (nodular cast iron) - by 0.85 and for machining group 38.1 (hardened steels, HRC 45 max) – by 0.55.

The feeds in Table 37 correspond to milling plain carbon and low alloy steels (groups 1-9) by inserts RC..T…MO. In case of milling material groups 10-11, the table feeds should be multiplied by 0.7; and for milling material groups 12-13 (ferritic and martensitic stainless steels) – by 0.85.

In either case, machining under unstable operational conditions will also demand reducing the feed by 30%. Using Table 36, the programmed starting feed for the previous example should be as follows: 0.4 (the table value) × 0.7 (coefficient for material group No.10) = 0.28 (mm/tooth).

Example

A workshop plans to perform a semi-finish machining of a deep mold cavity by using endmill cutter ERW D028A040-A-4-C32-12 with inserts RCCW 1206MO IC908. The cutter is clamped into a spring collet with overhang 150 mm. The workpiece material is unhardened AISI A2 tool steel. The programmed feed per tooth needs to be found if the cavity planned to be machined is with 0.6 mm depth of cut.

The machined material relates to group No.10.
d (the insert diameter) = 12 mm; and the ratio “depth of cut to insert radius”: ap/r=2×ap/d=0.1.
From Table 36 fzo=0.54 mm/tooth (for groups 5-9).
After correcting by the material coefficient (0.7) and the stability factor (0.7), the programmed feed will be: 0.54×0.7×0.7=0.26 (mm/tooth).

Table 36 **MILLSHRED Plain Round Inserts RC..W…MO. Basic Feeds, mm/tooth**

ap/r	Basic starting feed fzo, mm/tooth, for inserts					
	Ø12		Ø16		Ø20	
	ap	fzo	ap	fzo	ap	fzo
–	0.15	0.8	0.15	1	0.15	1.3
1/16	0.37	0.7	0.5	0.9	0.62	1.2
1/10	0.6	0.54	0.8	0.7	1	0.9
1/8	0.75	0.5	1	0.6	1.25	0.8
1/4	1.5	0.4	2	0.54	2.5	0.67
3/10	1.8	0.36	2.4	0.45	3	0.54
1/2	3	0.27	4	0.36	5	0.45
5/8	3.75	0.21	5	0.28	6.25	0.34
3/4	4.5	0.21	6	0.28	7.5	0.34
7/8	5.25	0.21	7	0.28	8.75	0.34
1	6	0.21	8	0.28	10	0.34

The feeds relate to workpiece material groups 5-9 (low alloy and quenched carbon steels) and 15-16 (grey cast iron).
For machining groups 10-11 (high alloy and tool steels), the table values should be multiplied by 0.7,
for machining groups 17-18 (nodular cast iron) - by 0.85 and
for machining group 38.1 (hardened steel, HRC 45 max) - by 0.55.
☐ - Recommended depths of cut

Table 37 **MILLSHRED Plain Round Inserts RC..T…MO. Basic Feeds, mm/tooth**

ap/r	Basic starting feed fzo, mm/tooth, for inserts					
	Ø12		Ø16		Ø20	
	ap	fzo	ap	fzo	ap	fzo
–	0.15	0.56	0.15	0.75	0.15	0.9
1/16	0.37	0.5	0.5	0.68	0.62	0.83
1/10	0.6	0.41	0.8	0.55	1	0.67
1/8	0.75	0.36	1	0.47	1.25	0.57
1/4	1.5	0.26	2	0.34	2.5	0.42
3/10	1.8	0.24	2.4	0.32	3	0.39
1/2	3	0.18	4	0.25	5	0.3
5/8	3.75	0.14	5	0.19	6.25	0.23
3/4	4.5	0.14	6	0.19	7.5	0.23
7/8	5.25	0.14	7	0.19	8.75	0.23
1	6	0.14	8	0.19	10	0.23

The feeds relate to workpiece material groups 1-9 (plain carbon and low alloy steels).
For machining groups 10-11 (high alloy and tool steels) the table values should be multiplied by 0.7, for machining groups 12-13 (ferritic and martensitic stainless steels) – by 0.85.
☐ - Recommended depths of cut

Tool overhang

The overhang (or the projection) of a milling tool is an important factor of the tool stiffness and machining stability. The tool overhang being 5% less reduces the tool deflection by 15%, 10% less –by 27% and 20% less – already by 50%. Minimizing the overhang substantially improves operational efficiency, allowing for increased cutting conditions and good surface finish.
But, what can we do - manufacturing real parts often demands long tools. How to determine the cutting data for such tools? And what is a high overhang, for which various techniques of cutting data determination recommend correction factors?
This not simple question is directly connected with the dynamic behavior of a tool. It relates to the sphere of serious research and needs a separate discussion. We are sure that the reader on the basis of his own knowledge and experience knows exactly if the overhang of a tool that he uses is high. For a rough estimate, the following rule of thumb often can be helpful: the overhang is high being 4-5 and more times as much as the nominal diameter of the tool. However, one thing needs clarification: from which point should the overhang be measured?
Generally, in case of the shell mills mounted on arbors, the correct way is to measure the overhang for the whole assembly, which is to say from the gauge line (datum) of the arbor shank (Fig. 9).
For the endmills that are clamped into holders with spring collets or adapter-style holders with side screws, the overhang is measured from the holder (Fig. 10).

ii. MILLSHRED serrated round inserts

As previously noted (Table 33), the serrated round inserts are intended first of all for the following cases:

- High depth of cut (above 15% of the insert diameter)
- Considerable tool projection (the overhang more than 2.5 of the tool diameter)
- Milling near thin walls
- Poor workholding

Additionally, the serrated inserts are sometimes applied in milling with usual depth of cut (to 15% of the insert diameter) when the main drive power of a machine tool is limited.

A programmed starting feed per tooth fz is defined by equation (8):

$$fz = fzo \times KH \qquad (8)$$

Where: fzo – the basic starting feed (Tables 39-41)
KH – the overhang coefficient (Table 38)

Fig. 9

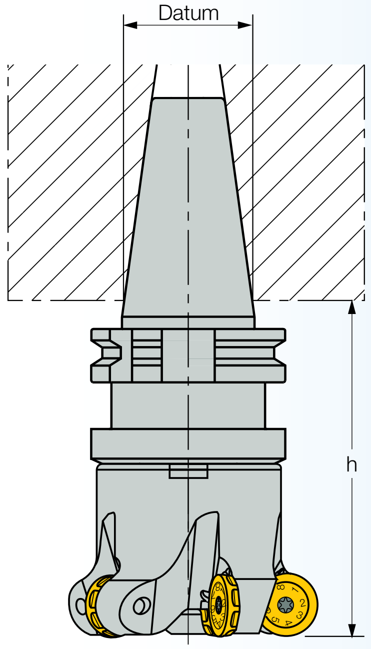


Fig. 10

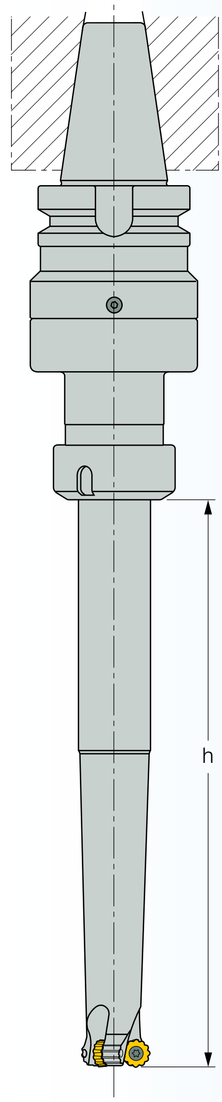


Table 38 **Overhang Coefficient KH for the MILLSHRED Serrated Round Inserts as the Function of Ratio of an Overhang H to a Tool Diameter D1**

H/D1	to 4	over 4 to 6	over 6 to 8	over 8 to 10
KH	1	0.85	0.7	0.65

Example

A die part made from AISI D2 tool steel, HB 210, is milled by cutter FRW D034A050-04-22-16 with inserts RCMT 1607-FW IC908 mounted on it. The cutter overhang is 120 mm, the depth of cut – 6 mm. What is a programmed feed to start milling?

The workpiece material relates to the tenth material group (No.10). Table 39 specifies the basic feed for a 16 mm diameter insert with 6 mm depth of cut as 0.2, which should be multiplied by 0.9 for material group No.10. The cutter diameter is 50 mm; and for overhang 120 mm the overhang coefficient is equal to 1 (Table 38).

Therefore we have fz = 0.2×0.9×1=0.18 (mm/tooth).

Table 39 **MILLSHRED General-purpose Serrated Round Inserts RCMT…FW. Basic Feeds, mm/tooth**

ap/r	Basic starting feed fzo, mm/tooth, for inserts					
	Ø12		Ø16		Ø20	
	ap	fzo	ap	fzo	ap	fzo
ap min	0.9	0.3	1.2	0.4	1.2	0.52
1/4	1.5	0.28	2	0.35	2.5	0.46
3/10	1.8	0.25	2.4	0.32	3	0.42
1/2	3	0.22	4	0.28	5	0.35
5/8	3.75	0.15	5	0.2	6.25	0.25
3/4	4.5	0.15	6	0.2	7.5	0.25
7/8	5.25	0.15	7	0.2	8.75	0.25
1	6	0.15	8	0.2	10	0.25

The feeds relate to workpiece material groups 1-9 (plain carbon and low alloy steels). For machining groups 10-11 (high alloy and tool steels) the table values should be multiplied by 0.9, for machining groups 15-18 (cast iron) – by 1.3 and for machining group 38.1 (hardened steel, HRC 45 max) – by 0.6.

☐ - Recommended depths of cut

Table 40 **MILLSHRED Serrated Round Inserts RCMT…FW-T20. First choice for machining martensitic stainless steel can also be used for machining carbon and low alloy steels**

ap/r	Basic starting feed fzo, mm/tooth, for inserts					
	Ø12		Ø16		Ø20	
	ap	fzo	ap	fzo	ap	fzo
ap min	0.9	0.26	1.2	0.35	1.2	0.4
1/4	1.5	0.23	2	0.3	2.5	0.35
3/10	1.8	0.21	2.4	0.28	3	0.3
1/2	3	0.18	4	0.25	5	0.28
5/8	3.75	0.13	5	0.18	6.25	0.21
3/4	4.5	0.13	6	0.18	7.5	0.21
7/8	5.25	0.13	7	0.18	8.75	0.21
1	6	0.13	8	0.18	10	0.21

The feeds relate to workpiece material groups 12-13 (ferritic and martensitic stainless steels). For machining groups 1-9 (carbon and low alloy steels) the table values should be multiplied by 1.1.

☐ - Recommended depths of cut

Example

A mold part from AISI 420 stainless steel with a thin-walled complicated shape and non-uniform machining allowance is milled by tool ERW D020A032-B-3-C32-12 carrying inserts RCMT 1206-FW-T20 IC928. Due to the unsteady allowance the depth of cut varies from 3 to 4 mm. The tool performs cutting with 220 mm overhang from a collet. Find feed per tooth needed for programming the tool path.

The material represents group No.14; ratio "overhang/diameter" is 220/32=6.9. Although Table 40 does not contain the basic feed for 12 mm diameter inserts with 4 mm depth of cut (worst case of loading) directly, it is easily seen that it is 0.13 mm.
Consequently, the starting programmed feed is: 0.13×0.7=0.09 (mm/tooth).

Table 41 **MILLSHRED Serrated Round Inserts RCMT…FW-F20.**
First choice for machining austenitic stainless steel and aluminum alloys; can be used also for machining low carbon steel.

ap/r	Basic starting feed fzo, mm/tooth, for inserts					
	Ø12		Ø16		Ø20	
	ap	fzo	ap	fzo	ap	fzo
ap min	0.9	0.13	1.2	0.17	1.2	0.2
1/4	1.5	0.12	2	0.16	2.5	0.18
3/10	1.8	0.1	2.4	0.14	3	0.16
1/2	3	0.09	4	0.12	5	0.14
5/8	3.75	0.07	5	0.09	6.25	0.1
3/4	4.5	0.07	6	0.09	7.5	0.1
7/8	5.25	0.07	7	0.09	8.75	0.1
1	6	0.07	8	0.09	10	0.1

The feeds relate to workpiece material group 14 (austenitic stainless steel).
For machining group 1 (low carbon steel) the table values should be multiplied by 1.5.
For machining groups 21-25 (aluminum alloys) the table values should be multiplied by 3.
☐ - Recommended depths of cut

b) Starting cutting speed Vc

The starting cutting speed should be determined by the following steps:

- Find the recommended cutting speed with the use of the averaged data in the tables
- Calculate the effective diameter
- Calculate the programmed spindle speed relative to the nominal diameter of a tool in such a way that the cutting speed relative to the effective diameter will be equal to the found value

Recommended cutting speed

$$Vc = Vo \times KF \times KH \times Kt \qquad (9)$$

Where:

Vo – The basic cutting speed (Table 43)

KF – The factor of a cutting edge form:
KF=1 for the plain round inserts and
KF=0.75 for the serrated round inserts

KH – The overhang coefficient:
For the serrated round inserts KH can be found from Table 38
For the plain round inserts KH is specified in Table 42

Kt– The tool life factor (Table 8)

Table 42 **Overhang Coefficient KH for the MILLSHRED Plain Round and the HELIDO H400 Round Inserts as the Function of Ratio of an Overhang H to a Tool Diameter D1**

H/D1	to 2.5	over 2.5 to 3	over 3 to 5	over 5
KH	1	0.9	0.65	0.5
Tool Application	Recommended		Less recommended	

In addition, for hard loading with broad-area engagement, improper workholding or milling thin-walled parts, the starting speed should be reduced by 20%.

Calculating the effective diameter De

Calculating the effective diameter De

$$De = D + 2 \times \sqrt{d \times ap - ap^2} \qquad (10)$$

Where:

- ***D*** – insert center circle
- ***d*** – insert diameter
- ***ap*** – axial depth of cut

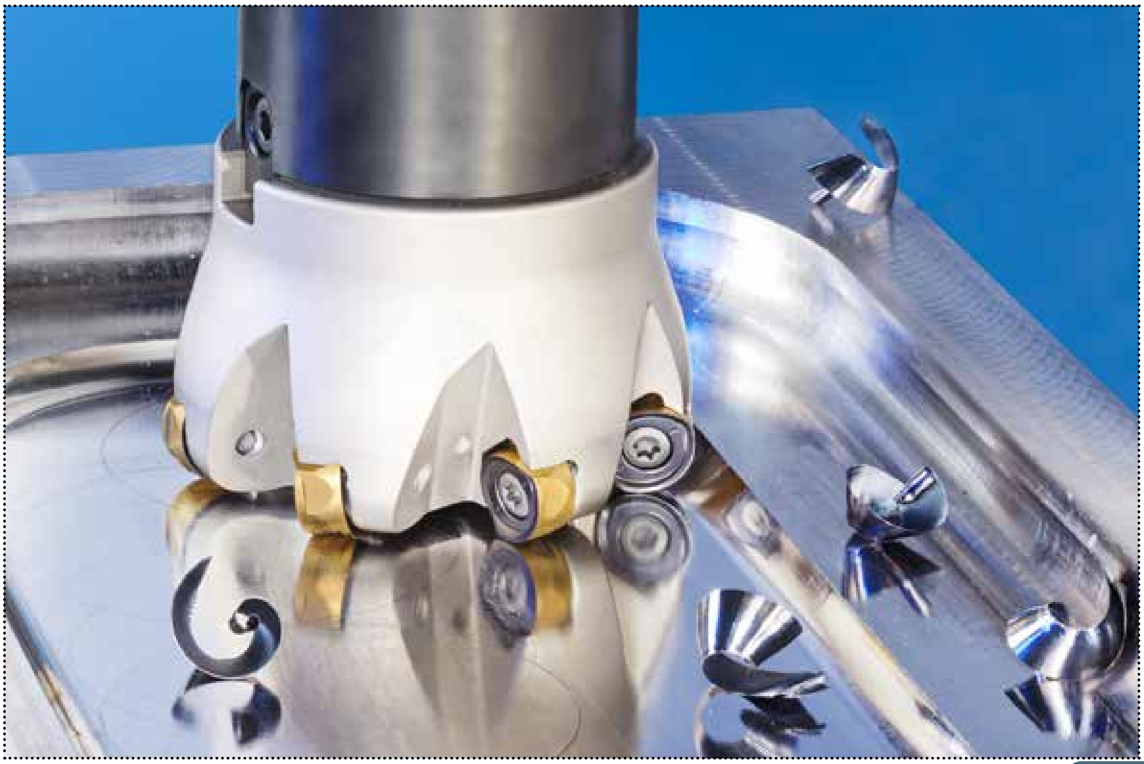
(For nominal diameter D1 of a tool D=D1-d)

Calculating the programmed spindle speed N

$$N = 1000 \times Vc/(\pi \times De) \qquad (11)$$

Cutting speed Vn relative to nominal diameter D1, can be found from the following equation:

$$Vn = \pi \times D1 \times N/1000 \qquad (12)$$



Effective diameter

The correct approach to finding the cutting speed of a profiling milling tool is to calculate the speed with respect to the effective diameter of the tool. In profiling, due to the shaped, non-straight form of the tool, a cutting diameter is a function of a depth of cut; and it is not the same for different areas of the tool cutting edge that is involved in milling. The effective diameter is the largest true cutting diameter: maximum of the cutting diameters of the mentioned areas (Fig. 11). Generally, it corresponds to the diameter measured at the axial depth of cut.

Ignoring the cutting diameter in calculating spindle speeds can cause essential errors in cutting data and incorrect tool operation. In the milling cutters with round inserts, specifically, this is particularly significant for relatively small tool diameters. For example, for a 25 mm diameter tool with 12 mm diameter round inserts, the cutting diameter varies from the value closed to 13 mm (the axial depth of cut is nothing more than tenths of mm) to 25 mm (at maximum depth of cut that equal to the insert radius – 6 mm). If the effective diameter is, for instance, 19 mm, and the spindle speed was calculated with reference to 25 mm, the real cutting speed relative to 19 mm diameter will be 24% less! The error grows for shallow depths of cut: for effective diameter 14 mm it is already 44%.
Of course, for large-sized tools such error grows substantially smaller and can even be ignored. Moreover, machining slots, milling near straight walls, etc. with even shallow depth of cut but with several passes, when after each pass the tool embeds deeper and deeper into the workpiece, and its nominal diameter in some or other way cuts the material (Fig. 12), the calculations shall relate only to the nominal diameter!
In any other case we always recommend to take the effective diameter into consideration and make corresponding necessary corrections during programming the spindle speed.
Remember, the cutting speed relates to the effective diameter, while the spindle speed refers to the nominal diameter!

Fig. 11

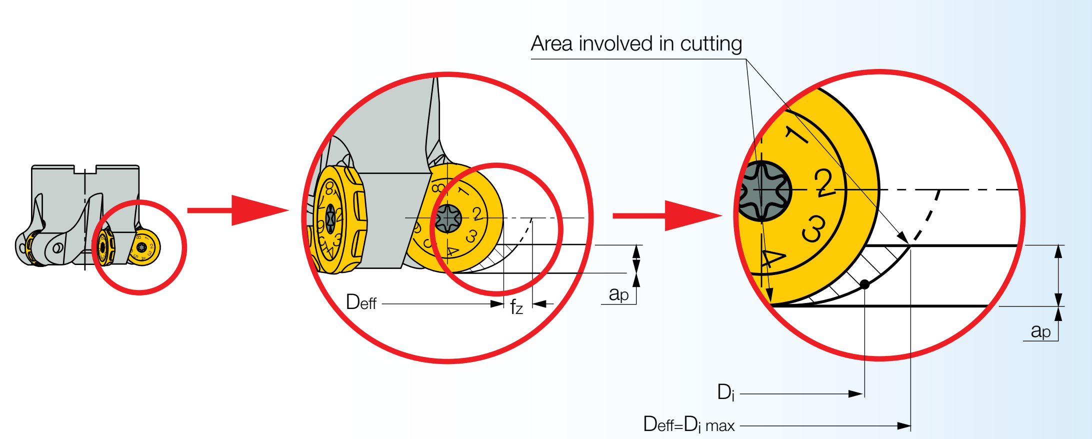


Fig. 12

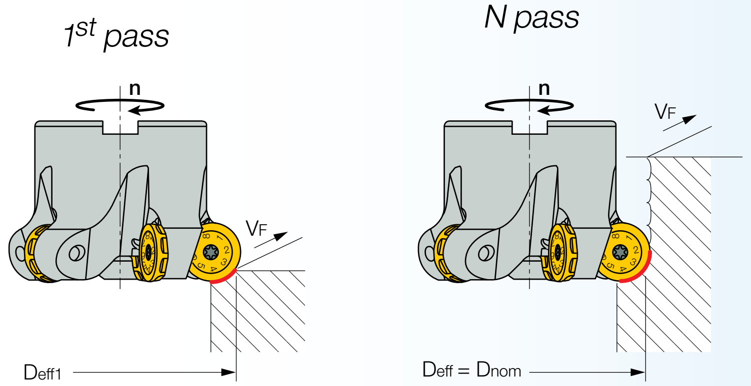


Table 43 **Basic Speed Vo, m/min, for MILLSHRED Round Inserts***

ISO Class DIN/ISO 513	Material Group**	ISCAR Carbide Grades		
		IC908	IC910	IC928
P	1	260	240	190
	2-4	240	210	170
	5	220	200	155
	6,7	210	180	155
	8-9	190	160	145
	10	190	145	135
	11	150	135	130
M	12, 13	180		155
K	15-16	240	270	230
	17-18	220	230	200
H	38.1***	110		70

☐ - First choice for grades

* For 20 min. tool life

** ISCAR material group in accordance with VDI 3323 standard

*** HRC 45 max.

Example

Find the cutting speed for one of the previous examples, in which the die part made from AISI D2 tool steel, HB 210, is milled by cutter FRW D034A050-04-22-16 carrying inserts RCMT 1607-FW IC908; while the cutter overhang is 120 mm and the depth of cut 6 mm. The starting programmed feed has been defined as 0.18 mm/tooth.

For material group No.10, that relates to AISI D2 steel Vo=190 m/min (Table 43).
KF=0.75 (for serrated inserts), KH=1 (overhang coefficient, Table 38).
Hence for 20 min. tool life Vc=190×0.75×1=143 (m/min)

The effective diameter for 6 mm depth of cut: $De=34+2\times\sqrt{(16\times6-0.6^2)}=49.5$ (mm).
The received value is very close to the tool nominal diameter (50 mm). Therefore in calculating the spindle speed the latter can be used because the errors will be negligible.
The programmed spindle speed: $N=1000\times143/(\pi\times50)=910$ (rpm)
Tool life factor Kt=0.8 for 60 min. tool life expectancy (Table 8), and in for this case the cutting speed and the spindle speed should be 114 m/min and 728 rpm respectively.

Example

Find the cutting data for milling a plate from pre-hardened mold steel AISI P20, HRC 32, with the use of shell mill FRW D080A100-06-32-20 with inserts RCMW 2009MO IC928, if the depth of cut is 1.8 mm and the tool overhang 86 mm.
If the ratio “overhang to tool diameter”, 86/100, is small, then KH=1 (Table 42).
The effective diameter: $De=80+2\times\sqrt{(20\times1.8-1.8^2)}=91$ (mm).
KF=0.75 (for plain round inserts).
The workpiece material in its condition corresponds to material group No. 9.
The starting programmed feed is found from Table 36: fz≈0.73 mm/tooth (the value between table values 0.8 and 0.67).
The recommended cutting speed: Vc=145×1×1×1=145 (m/min).
The programmed spindle speed: $N=1000\times145/(\pi\times91)=507$ (rpm).
The cutting speed with respect to the tool nominal diameter:
$Vn=\pi\times100\times507/1000=160$ (m/min)

Example

A customer complains of poor tool life with inserts RCCW 1206MO IC908, that are used in milling a deep and narrow cavity with almost vertical walls from a solid block of SAE/AISI 4340 steel. During operation, tool ERW D020A032-A-3-C25-12, with the inserts clamped into its pockets, machines the cavity by the required contour with shallow cuts (0.7 mm depth of cut) by stepped passes and thus removes material layer by layer. The customer defined the cutting data in accordance with the ISCAR recommendations and took into account the high overhang and the effective diameter. What is a reason for the problem?

In the first pass only a small part of the insert cutting edge cuts the material. But every next pass brings a new portion of the edge in engagement. After already the 8th pass, almost all cutting portions will be involved in milling near the cavity wall: 8×0.7 mm =5.6 mm, and the maximal depth of cut is 6 mm – the insert radius.

The calculated effective diameter is $20+2\times\sqrt{(12\times0.7-0.7^2)}=25.6$ (mm); and with respect to this value the customer defined the spindle speed. As a result, the cutting speed referred to the nominal diameter (that began cutting after 8 passes) is 1.25 (32/25.6=1.25) greater than required. In all appearances, it is the reason for the customer's problem. Even though the depth of cut is small, the tool nominal diameter cuts also and therefore the calculation should be relative to it.

In summation, the above-mentioned advantages: high strength of cutting edge and benefits of the round shape, made the round insert the real workhorse of rough and semi-finish milling in die and mold making. Marking the end of the section, underlined again are some points of using MILLSHRED round inserts:

- For shallow depths of cut and small tool overhang, the plain round inserts are the best solution. Their highest efficiency will be when the depth of cut is 10-30% of the insert radius (or 5-15% of the insert diameter). Remember that machining with deeper cuts in this case is a source of serious vibrations, intensive wear and misoperation.
- Machining with greater depths of cut, high overhang, insufficient stiffness of the technological system (poor workholding, thin-walled part, etc.) will be more effective with the serrated round inserts.
- The serrated inserts due to chip splitting effect can solve the problem of chip evacuation.
- The serrated inserts need less cutting power and therefore are suitable for low-power machine tools.
- Cutting data must be correctly defined with respect to properly chosen effective diameter and depth of cut, because the wrong definition can lead to substantially reduced tool life.

3.1.1.1. HELIDO H400 Round Line
A Conceptually New Solution for Efficient Profiling

The **HELIDO** H400 family, one of the latest ISCAR developments, was designed in order to fulfill the industry requirement for an effective milling tool for multiaxis profiling. The family is based on a double-sided insert H400 RNHU 1205… that looks like a round insert, but properly speaking is not round in the sense of the word.
The H400 RNHU 1205… insert (Fig. 13) has four major cutting edges and each of them is the 120° arc of 6 mm radius. By comparison, a round insert of traditional design ensures four 90° arcs. In addition, the insert has four minor cutting edges, acting mostly during ramp down.
The four full 120° major cutting edges in combination with the four "ramping" edges significantly expand the family possibilities in milling complicated 3-D shapes, especially in 5-axis machining, (Fig. 14) that makes it promising for die and mold applications.
The H400… insert is smaller in size, that enables a cutter design with more insert at the periphery – a more closed angular pitch of teeth.
The **HELIDO** H400 cutters are available in arbor-type face mill, shank-type endmill configuration; and also as interchangeable milling heads with **FLEXFIT** threaded connection. The range of diameters for the cutters of the standard line is 32…80 mm.
There are three different types of the insert: ML, HP and AX, which are designed for machining various materials and applications. The inserts of ML- and HP- types have straight major edges, the inserts of AX-type – helical (Fig. 15). The inserts are made from the latest SUMO TEC carbide grades, featuring advanced coating and post treatment technology. Table 44 shows general data regarding the inserts and their application in the die and mold industry. As seen, the inserts of ML- and AX-types are more suitable for milling the die and mold materials. At the same time the inserts of HP-type having so-called "high positive" cutting geometry and basically intended for high temperature alloys, can also be used in die and mold practice, so far as milling soft carbon steel or some kinds of martensitic stainless steel.

Fig. 13

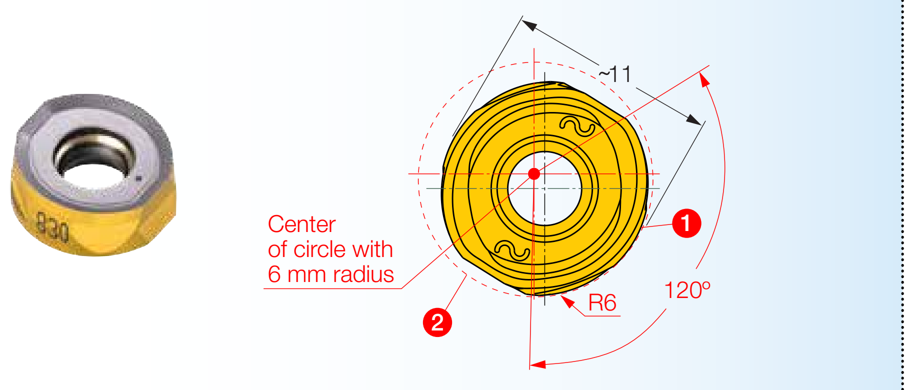
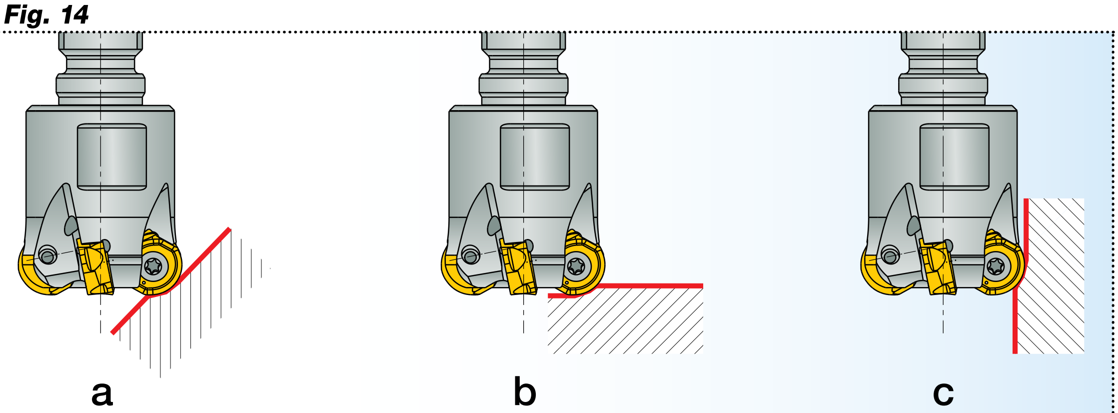


Fig. 14



Milling Tools

Die and Mold

Fig. 15

Straight Edge **ML, HP** Helical Edge **AX**

a **b**

Neutral, positive and high positive

In milling, unlike turning, there is no standard classification for combinations of geometrical parameters of indexable inserts, but words “neutral”, “positive”, etc. are commonly used. In order to be familiar with the terms they need clarification.
An insert with a flat rake surface parallel to the insert base is called “neutral”.
Typical examples of such a geometry are inserts for milling hardened steel and cast iron.
But what is “positive”?
The progress in powder metallurgy and pressing technology brought about the possibility of shaping the rake face, and the possibility became more and more impressive. The shaping resulted in an effective way of increasing positive normal rake and axial rake for indexable milling tools, which led to substantially decreased cutting forces, improving chip formation and making cutting smoother and more stable. This is the reason why that sort of milling inserts is called “positive”.
Further development of technology gave rise to an additional increase of difference in height for two cutting corners along the insert cutting edge. Thus the axial rake can be more positive.
It is accepted to call the inserts with the considerable height differences “high positive”.
Of course, the pressing technology does not exhaust its potential, and maybe, today’s high positive milling inserts will be classified only as positive tomorrow.

Table 44 **HELIDO H400 Inserts**

Insert	Cutting edge	T-land	Intended for		Grades		
			First priority	Second priority	IC808	IC830	IC330
H400 RNHU...-ML	straight	negative	steel	stainless steel			
H400 RNHU...-AX	helical	neutral	stainless steel	steel			
H400 RNHU...-HP	straight	positive	high temp. alloys	stainless and soft steels			

The way of defining starting cutting data for the H400 milling tools is similar to the already described method relative to the cutters with indexable round inserts. Equation (6) specifies a starting feed, equation (7) – a chip thinning factor and (9) – a starting cutting speed. Cutting edge angles can be calculated by equations (4)-(5a) or estimated using Table 32 (in this case r will be the radius of a H400 insert: namely, 6 mm for inserts H400 RNHU 1205). The tables below show values of the basic starting feed fzo and speed Vo (Tables 45 and 46 correspondingly) and allow estimation of the basic feed directly (Table 47).

Table 45 **HELIDO H400 RNHU 1205 Round Inserts. Basic Starting Feed fzo, mm/tooth**

ISO Class (DIN/ISO 513)	Material Group*	fzo for insert type		
		ML	AX	HP
P	1-4	0.21	0.14	0.14
	5-9	0.21	0.14	
	10-11	0.15	0.1	
M	12-13	0.18	0.12	0.12
K	15-16	0.21		
	17-18	0.18		
H	38.1	0.12		

* ISCAR material group in accordance with VDI 3323 standard

☐ - Recommended application

Table 46 **Milling Tools with HELIDO H400 RNHU 1205 Inserts. Basic Speed Vo, m/min**

ISO Class DIN/ISO 513	Material Group**	Vo for carbide grades		
		IC808	IC830	IC330
P	1	260	180	160
	2-4	240	160	145
	5	220	145	130
	6,7	210	145	130
	8-9	190	140	125
	10	190	135	125
	11	150	125	120
M	12, 13	180	150	140
K	15-16	240	240	
	17-18	220	220	
H	38.1***	110	90	

☐ - First choice for grades

* For 20 min. tool life

** ISCAR material group in accordance with VDI 3323 standard

*** HRC 45 max

Table 47 **HELIDO H400 RNHU 1205 Round Inserts. Basic Feeds fzo, mm/tooth, as a Function of Depth of Cut ap**

ap, mm	fzo for types		
	ML*1	AX*2	HP*3
0.15	0.8	0.56	0.56
0.37	0.7	0.5	0.5
0.6	0.54	0.41	0.41
0.75	0.5	0.36	0.36
1.5	0.4	0.26	0.26
1.8	0.36	0.24	0.24
3	0.27	0.18	0.18
3.75	0.21	0.14	0.14
4.5	0.21	0.14	0.14
5.25	0.21	0.14	0.14
6	0.21	0.14	0.14

*1 ML-type:
The feeds relate to workpiece material groups 1-9 (plain carbon and low alloy steels) and 15-16 (grey cast iron). For machining groups 10-11 (high alloy and tool steels) the table values should be multiplied by 0.7, for machining groups 12-13 (martensitic stainless steel) and 17-18 (nodular cast iron) - by 0.85 and for machining group 38.1 (hardened steel, HRC 45 max) - by 0.55.

*2 AX-type:
The feeds relate to workpiece material groups 1-9 (plain carbon and low alloy steels).
For machining groups 10-11 (high alloy and tool steels) the table values should be multiplied by 0.7, for machining groups 12-13 (ferritic and martensitic stainless steels) - by 0.85.

*3 HP-type:
The feeds relate to workpiece material groups 1-4 (plain carbon steel).
For machining groups 12-13 (ferritic and martensitic stainless steels) the table values should be multiplied by 0.85.

Concerning correction caused by a tool overhang, Table 42 gives necessary data.

Example
A die and mold manufacturer decided to use endmill cutter H400 ER D32-4-060-C32-12 with inserts H400 RNHU 1205-HP IC830 for 5-axis machining a complex-shaped mold component made from AISI 420 stainless steel with hardness HB 300…310. The depth of cut during machining varies from 1 to 4.5 mm. Machining stability is estimated as good. The manufacturer approached a local ISCAR representative on recommendations regarding starting cutting data.

The component material relates to ISCAR material group No.13. Due to variable depth of cut the calculation should refer to the depth maximum (4.5 mm).

The effective diameter: $De=(32-12)+2\times\sqrt{(12\times4.5-4.5^2)}=31.6$ (mm)$\approx$32 mm (the most part of the cutting edge performs cutting).

The basic feed per tooth (Table 45): fzo=0.18 mm/tooth (alternatively, fzo can be defined from Table 47 with reducing the table value by 15% - see remark to the table).
The basic cutting speed (Table 46): Vo=150 m/min

The programmed starting spindle speed: $N=1000\times150/(\pi\times32)=1492$ (rpm).
The starting feed speed: $V_F=0.18\times4\times1492=1074$ (mm/min).

3.1.2. MULTI-MASTER Toroidal Milling Heads and Solid Carbide Toroidal Milling Tools

As for other forms of the **MULTI-MASTER** heads, the toroidal cutting profile if this tool family is also represented by two design versions: two-flute pressed to shape and size MM HT… heads and multiflute MM ETR… heads fully ground from solid.
Table 48 shows nominal diameters D and corner radii R of the standard line of the **MULTI-MASTER** toroidal heads.

Table 48 **MULTI-MASTER Toroidal Milling Heads**

Tool diam. D, mm	8	10				12					16				20				
Corner radius R, mm	**2**	**0.5**	**1**	**2**	**3**	**1.6**	**2**	**2.5**	**3**	**4**	**2**	**3**	**4**	**5**	**3**	**4**	**5**	**6**	**8**
2 flute MM HT...																			
6 flute MM ETR...																			

Back draft of toroidal cutting bull

If the diameter of a toroidal milling tool with round inserts decreases it causes reduction of a screw that secures the insert. The tool becomes weaker, less productive; and at one point producing such tools is practically impossible. The same situation occurs when the insert radius decreases. However, the die and mold industry demands toroidal cutters to be able to effectively mill light-size parts and small radii. In addition, increased requirements of machining accuracy set a limit to using the tool with round inserts in some operations.

Solid carbide tools and interchangeable heads with a toroidal cutting profile overcome these problems and thus extend the application boundaries of toroidal milling tools.

There are some technical terms, which specify the solid tools and the heads. Because the tool manufacturers and industrial workers often use different terms for the same things, the matter requires short consideration.

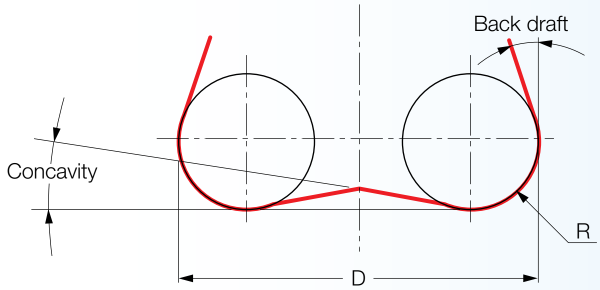
Speaking about the 90˚ solid endmill, they are traditionally divided between the square endmills with sharp corners, the radiused (or radius) endmill with rounded corners and the chamfered end mill with chamfered corners. The cutting end of a tool (on the tool face) is made with concavity or without it (in flat bottom endmills); and the peripheral cutting end may or may not have a back taper. The back taper allows for minimization of the area of contact between the tool and a machined wall if this contact is undesirable (the tool deflection due the radial cutting force, high overhang, vibrations, etc.). If truth be told, even the flat bottom endmills are produced with small concavity, the straight peripheral cutting edge often have slight, and the sharp corners upon closer view are not so sharp but gently rounded or chamfered; but we do not take these conventional design features into consideration under current discussion.

So, the radiused, also called the bull nose, mills, especially with the concave end cutting edge, can perform as the toroidal cutters. A back draft of the peripheral cutting edge, produced by the back taper, improves efficiency of the cutter.

Today solid carbide toroidal endmills and interchangeable toroidal milling heads feature the design configuration of the back draft radiused (bull nose) cutters. The most popular diameters of the tools and heads lay within 10…25 mm. Normally they have 5˚...7˚ concavity and back draft; and the number of teeth is generally 2 or 3. Being ground peripherally and mostly with ground rake face, the tool and heads combine the advantages of the round inserts with the precise radiused endmills and are suitable for rich applications from productive milling of small die and mold components to finish fillets with high accuracy. In some cases these tools and heads are provided with bottom or peripheral wiper flats and even chip splitting grooves.

Properly speaking, the toroidal solid tools and heads present the milling cutters with precise round inserts, where the insert is an integral part of the cutter body (Fig. 16).

Fig. 16



The main design features particular to MM HT… heads are: high-strength structure that allowed heavy tooth loading and neutral ground rake face (zero radial rake). These characteristic properties make the heads very effective in milling steel, hardened steel and cast iron. The heads are successfully applied to high feed milling, machining cavities and pockets by helical interpolation and ramping, side plunging, and also drills small depth.

Multiflute MM ETR… heads with 30° helix and zero radial rake are intended first of all for high-production finish milling of hardened steel and precise fillet finishing. The heads do not have center cutting teeth and therefore are not suitable for drilling operations. However, their geometry enables high-performance ramping down by straight line and by helix (interpolation).

In addition, both types of the heads are sometimes used for milling undercuts.

IC908 is a main carbide grade for the **MULTI-MASTER** toroidal milling heads. In addition, some heads that are intended mostly for milling steels of high hardness (HRC 56-63) are produced from grade IC903.

STARTING CUTTING DATA

a) Starting feed per tooth fz

Programmed feed per tooth fz, which relates to the nominal diameter of a head is defined by the above equation (8):

$$fz = fzo \times KH$$

Table 49 shows basic starting feed fzo for different groups of engineering materials.
If the overhang of a tool does not exceed 3D (three diameters of the tool), KH=1; in case of the greater overhangs KH=0.7.

In milling with the toroidal heads, the machining allowance is normally 0.01D…0.2D. If the heads are used, for example, for face milling, an axial depth of cut defines the allowance; and for heads with relatively large corner radius, the chip thinning effect may play an important role. The “relatively large corner radius” has no precise definition, but a rule of thumb says that it is the radius from 3 mm (2 mm for 8 mm dia. heads) and more. Table 50 contains values of chip thinning factor KTH that should be taken into account in such cases, for which

$$fz = fzo \times KH \times KTH \qquad (13)$$

b) Starting cutting speed Vc

Table 51 gives averaged starting cutting speeds. If the cutting conditions can be estimated as unfavorable (poor workholding, high tool overhang, milling thin walls) the table values should be reduced by 20-30%.

Table 49 **MULTI-MASTER Toroidal Heads: Basic Feed fzo, mm/tooth, for Head Diameter D, mm**[*1]

ISO Class DIN/ISO 513	Material Group[*2]	fzo for D				
		8	**10**	**12**	**16**	**20**
P	1-4	0.13	0.14	0.16	0.18	0.2
	5	0.12	0.13	0.15	0.16	0.18
	6, 7	0.1	0.11	0.12	0.13	0.15
	8, 9	0.09	0.1	0.12	0.13	0.15
	10	0.08	0.09	0.1	0.11	0.13
	11	0.07	0.08	0.09	0.1	0.11
M	12, 13	0.08	0.09	0.1	0.11	0.13
K	15-16	0.13	0.14	0.16	0.18	0.2
	17-18	0.12	0.13	0.15	0.17	0.18
H	38.1[*3]	0.06	0.06	0.07	0.07	0.08
	38.2[*4]	0.05	0.05	0.06	0.06	0.07
	39[*5]	0.03	0.03	0.04	0.04	0.05

[*1] Machining allowance (0.01…0.2) D for MM HT… heads and (0.01…0.06) D for MM ETR… heads
[*2] ISCAR material group in accordance with VDI 3323 standard
[*3] HRC 45-49
[*4] HRC 50-55
[*5] HRC 56-63
☐ – Most recommended applications

Toroid for improved surface quality in milling die and mold cavities

For better surface finish, ISCAR offers the customer toroidal heads with chip splitting grooves and side wiper flats (Fig. 17 and 18).
The splitting grooves allow producing smaller chips and thus reduce scratching the walls of a machined wall cavity. In addition, the smaller chip, swarf, makes chip evacuation easier.
The side wiper flat intended for better surface finish of the cavity walls. In order to achieve the best results in this case a stepdown should not exceed the length of the wiper flat

Fig. 17

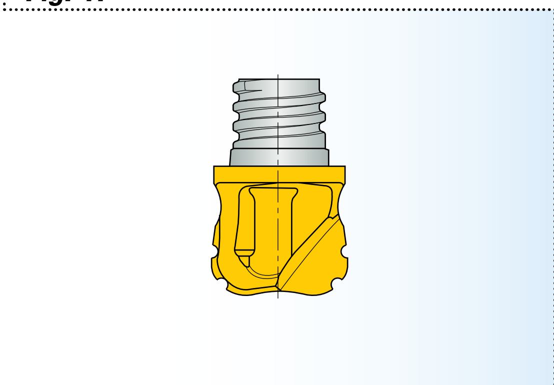


Fig. 18

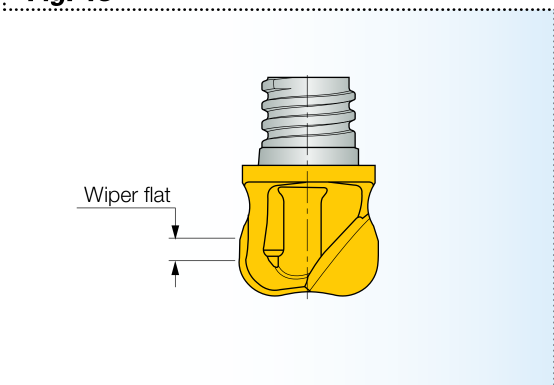


Table 50 **Chip Thinning Factor KTH for MULTI-MASTER Toroidal Heads with Nominal Diameter D, mm, and Corner Radius R*, mm**

ap/D	D=8		D=10		D=12			D=16				D=20					
	ap	**R2**	**ap**	**R3**	**ap**	**R3**	**R4**	**ap**	**R3**	**R4**	**R5**	**ap**	**R3**	**R4**	**R5**	**R6**	**R8**
0.01	0.08	3.6	0.1	3.5	0.12	3.2	3.6	0.16	2.8	3.2	3.5	0.2	2.4	2.9	3.2	3.5	4
0.02	0.16	2.5	0.2	2.5	0.24	2.2	2.2	0.32	2	2.2	2.5	0.4	1.8	2.1	2.2	2.5	2.9
0.05	0.4	1.7	0.5	1.6	0.6	1.4	1.7	0.8	1.3	1.4	1.6	1	1.2	1.3	1.4	1.6	1.8
0.1	0.8	1.2	1	1.2	1.2	1.1	1.3	1.6	1	1.1	1.2	2	1	1	1.1	1.2	1.3
0.15	1.2	1.1	1.5	1	1.8	1	1.1	2.4	1	1	1	3	1	1	1	1	1.1
0.2	1.6	1	2	1	2.4	1	1.1	3.2	—	1	1	4	—	1	1	1	1

* Machining allowance (0.01…0.2) D for MM HT… heads and (0.01…0.06) D for MM ETR… heads

It will be noted that a programmed spindle speed relates to the nominal diameter of a head. So, as in the case of the tools with the indexable round inserts, the spindle speed is tightly correlated with effective diameter De of a head in an application.

$$De = D - 2R + 2 \times \sqrt{(2R \times ap - ap^2)} \qquad (14)$$

Where:

D – the nominal diameter of a head
R – the head corner radius
ap – axial depth of cut

This is the correct way is to always define the effective diameter. However, small corner radii, such as 0.5 or 1 mm allow for neglecting it because the difference between the nominal diameter and the effective diameter has slight deflection. The same situation occurs when the cutting depth is close to the corner radius.
For example, the effective diameter of the head of 12 mm nominal diameter with 3 mm corner radius, which performs milling with 2 mm depth of cut, is 11.6 mm. For 0.5 mm depth it will be 9.3 mm, and for 0.3 mm depth – only 8.6 mm. It is clear, that calculations corresponding to the nominal diameter without correlation cause only a 3% error in the first case, 23% in the second, and almost 30% in the third.

A rule of thumb

If the difference between the nominal and the effective diameters is less than 20%, the cutting speed can be calculated with respect to the nominal diameter, and no correlation needs.

Table 51 **MM HT… Heads and HTR… Inserts: Starting Speed Vc, m/min**

ISO Class DIN/ISO 513	Material Group*	Vc, m/min
P	1	180
	2-4	160
	5-6	150
	7-9	140
	10	130
	11	120
M	12, 13	120
K	15-16	200
	17-18	180
H	38.1	120
	38.2	100
	39**	80

* ISCAR material group in accordance with VDI 3323 standard
** In this case HSM recommended
☐ – Most recommended applications

Example
A planner decided to use endmill cutter MM S-A-L065-W16-T08 with toroidal head MM HT120N06R3.0-2T08 908 for milling a part made from grey cast iron. The depth of cut was defined as 0.6 mm and the stepover as 10 mm. What is the programmed cutting data for proper workholding and stiffened tool clamping?

Cutting speed Vc=200 m/min (Table 51), basic feed fzo=0.16 mm/tooth (Table 49). Chip thinning factor KTH=1.4 (Table 50) and the programmed feed per tooth should be 0.22 mm/tooth (equation (13)).
Effective diameter $De=12-2\times3+2\times\sqrt{(2\times3\times0.6-0.6^2)}=9.6$ (mm); hence the programmed spindle speed should be $1000\times200/(\pi\times9.6)=6630$ (rpm).
Compare: if we relate directly to the nominal diameter 12 mm, the result might be 5305 rpm, and the real cutting speed transforms to 160 m/min – 20% less.

Feed speed $V_F=0.22\times2\times6630=2917$ (mm/min).

As previously stated, multiflute heads MM ETR… (and also the solid carbide endmills of similar cutting geometry) are most advantageous for finish milling hardened steel, more exactly with hardness to HRC 55. Machining allowance in this case does not exceed 0.06 D (normally even 0.03 D). Under such conditions the starting feed per tooth, the thinning factor and the starting cutting speed are the same as for MM HT… heads and therefore can be chosen from Tables 49-51. In finish milling with small allowance the cutting speed is actually 10-15% more than the table data.

Example
An insert of a plastic mold cavity is finished by endmill cutter MM S-A-L090-C12-T08 with head MM ETR080A04R2.0-6T05 908 clamped into it. The depth of cut is 1…1.3 mm, the stepover – 0.3 mm. Find the starting cutting data if the insert material is AISI P20 steel, and its hardness as of the operation is HRC 50…52.

The basic feed (Table 49) is 0.05 mm/tooth, the chip thinning factor (Table 50) – 1.1. Without any information regarding stability of machining we can, however, assume that it is sufficient (the short shank of type A, small allowance) and the programmed starting feed will be 0.05×1.1=0.055 (mm/tooth).

The starting cutting speed 100 m/min, which is taken from Table 51, can be even increased by 10-15%, as above, and is set 110 m/min.
Effective diameter $De=8-2\times2+2\times\sqrt{(2\times2\times1.3-1.3^2)}=7.7$ (mm). The effective diameter is very close to the nominal diameter of the head (96%); and corrections for a programmed spindle speed are not needed. The spindle speed 1000×110/(π×8)=4377 (rpm), the feed speed 0.055×6×4377=1444 (mm/min).

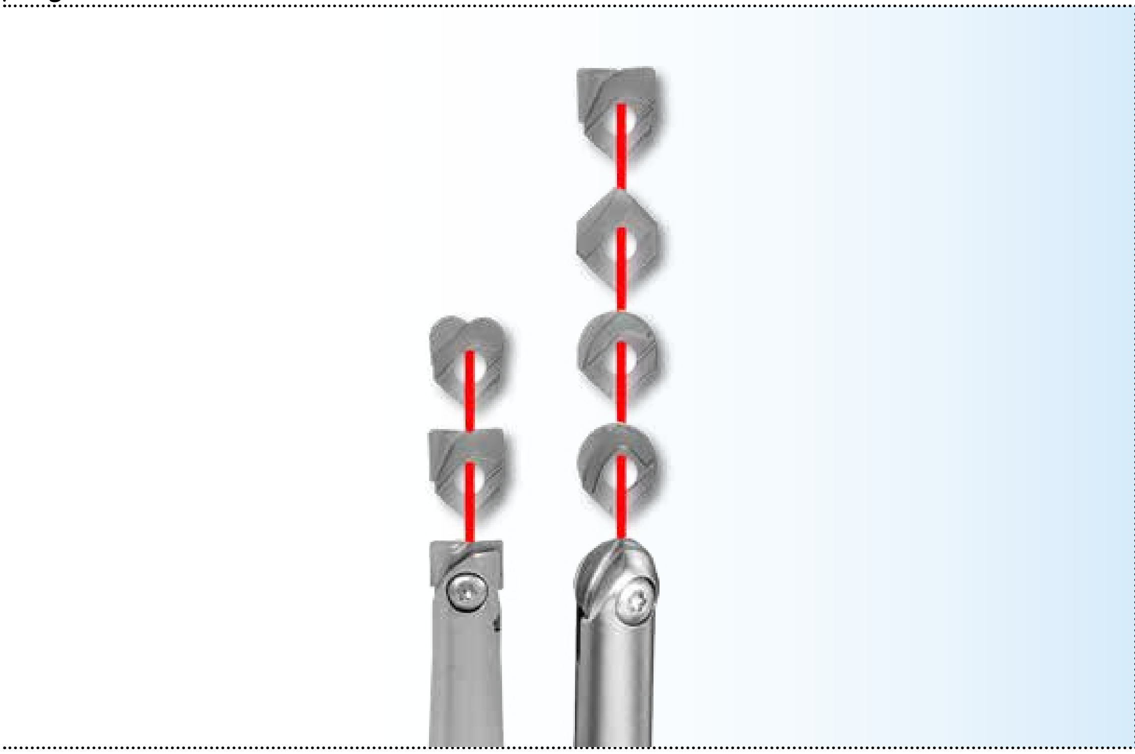
3.1.3. BALLPLUS one-insert toroidal milling tools

BALLPLUS (Fig. 19) is a multifunction system comprising straight and tapered cylindrical shank options and inserts of various types. A one-insert tool is a combination of an insert, which has cutting teeth and a shank. The shank has the ability to carry every type of insert. The insert rear (non cutting) V-shaped part is positioned against two contact surfaces within the corresponding V-formed slot of the shank; and a clamping screw being tightened pulls the insert towards the contact surfaces - that guarantees high accuracy and firm and secure insert mounting, even under considerable cutting forces. Replacing the inserts is very simple.

The **BALLPLUS** system offers the following insert shapes:

- Spherical and ball nose (hemispherical)
- Toroidal
- Square
- V-shaped (for chamfering and countersinking)

Fig. 19



Versatility of cutting tool (1)

Versatility or multifunctionality of an indexable cutting tool depends largely on a variety of cutting geometries that the tool provides. Using replaceable cutting heads or inserts of the same form but adopted for machining different workpiece materials by means of changed rake face and flank is a common practice. The tool becomes more and more versatile if it can carry inserts or heads of various forms.

In milling, the tool systems with replaceable heads have great possibilities for versatility when one tool body allows mounting heads of substantially different shape. For example, in the MULTI-MASTER family, combining the same shank with miscellaneous cutting heads produces effective tools from square endmills to ball nose cutters or from center drills and combined countersinks to disk slotting cutters. The one-insert tool systems enable fewer options for such combinations but also can be a good basis for versatile tooling. A shank of the ***BALLPLUS*** *system with an insert clamped into it performs as a ball nose or toroidal endmill or a countersink. Even milling tools with indexable inserts, for which the combining possibilities are very limited, nevertheless hold out reserve for increasing versatility. The already discussed* ***HELIDO*** *S845 family with its ability to carry either square or octagonal inserts is a good example of this.*

The versatile tools considerably increase the range of tool use and allow for reduction of a tool stock. Therefore, maximal versatility is an important design principle that forms the basis of high efficiency applications for milling tool families, especially those with replaceable cutting heads.

Multifunctionality based on combining the shank with the inserts of different shapes makes the **BALLPLUS** tools basically similar to the **MULTI-MASTER** family, by the virtue of main design principles.

Plunged bull

In addition to their main applications many toroidal tools are suitable for plunge with work feed along a tool axis, peck drilling and also direct drilling on a small depth.

Table 52 shows the range of corner radii with reference to the nominal diameter of the **BALLPLUS** toroidal milling inserts.

Table 52 **BALLPLUS Toroidal Milling Inserts**

Corner radius R, mm	Insert diam. D, mm			
	12	16	20	25
1				
1.5				
2				
3				
4				
5				
6				

STARTING CUTTING DATA

In general, starting cutting data for the **BALLPLUS** toroidal tools is defined in the same way as for the **MULTI-MASTER** toroidal mills. Again, the similar considerations take place for the cases when correlation due to chip thinning or substantial difference between a nominal and an effective diameter should be taken into account.

Equations (8) and (13) specify programmed feed per tooth fz. Table 53 contains values of basic starting feed fzo needed for calculations. Table 54 shows appropriate values of the chip thinning factor.

In rough and semi-finish milling, if the **BALLPLUS** toroidal tools machine plain surfaces, the width of cut is normally (0.6…0.7)×D, where D - the diameter of a tool, and the depth of cut – (0.08…0.15)×D (needless to say that in any case the depth of cut shall not exceed the corner radius of the applied insert). Under such conditions the tool performs with high productivity and ensures good insert life.
In finish milling, the machining allowance is usually no more than 0.12×D for pre-hardened steel, 0.08×D for steel with hardness HRC 45…49 (material group 38.1), 0.06×D for HRC 50…55 (group 38.2) and 0.05×D for HRC 56…63 (group 39).

Table 53 **BALLPLUS Toroidal Tools: Basic Feed fzo, mm/tooth**

ISO Class DIN/ISO 513	Material Group*	fzo for D, mm			
		12	16	20	25
P	1-4	0.18	0.19	0.21	0.23
	5	0.16	0.17	0.19	0.21
	6, 7	0.14	0.15	0.16	0.18
	8, 9	0.13	0.14	0.15	0.16
	10	0.11	0.12	0.13	0.14
	11	0.1	0.11	0.12	0.13
M	12, 13	0.11	0.12	0.13	0.14
K	15-16	0.18	0.19	0.21	0.23
	17-18	0.16	0.17	0.19	0.21
H	38.1	0.07	0.07	0.08	0.1
	38.2	0.06	0.06	0.07	0.08
	39	0.04	0.04	0.05	0.06

* ISCAR material group in accordance with VDI 3323 standard

As is the case with the **MULTI-MASTER** toroidal heads, Table 51 is intended for rating starting cutting speed Vc. In the same manner, the values should be reduced by 20-30% if the operation conditions are estimated as unstable.

Table 54 **Chip Thinning Factor KTH for BALLPLUS Toroidal Inserts with Nominal Diameter D and Corner Radius R**

ap/D	D=12			D=16		D=20			D=25				
	ap	R3	R4	ap	R3	ap	R3	R4	ap	R3	R4	R5	R6
0.01	0.12	3.2	3.6	0.16	2.8	0.2	2.4	2.9	0.25	2.2	2.6	2.9	3.1
0.02	0.24	2.2	2.2	0.32	2	0.4	1.8	2.1	0.5	1.6	1.9	2.1	2.2
0.05	0.6	1.4	1.7	0.8	1.3	1	1.2	1.3	1.25	1.1	1.2	1.3	1.5
0.1	1.2	.1	1.3	1.6	1	2	1	1	2.5	1	1	1	1.1
0.15	1.8	1	1.1	2.4	1	3	1	1	3.75	1	1	1	1
0.2	2.4	1	1.1	3.2	–	4	–	1	5	–	1	1	1

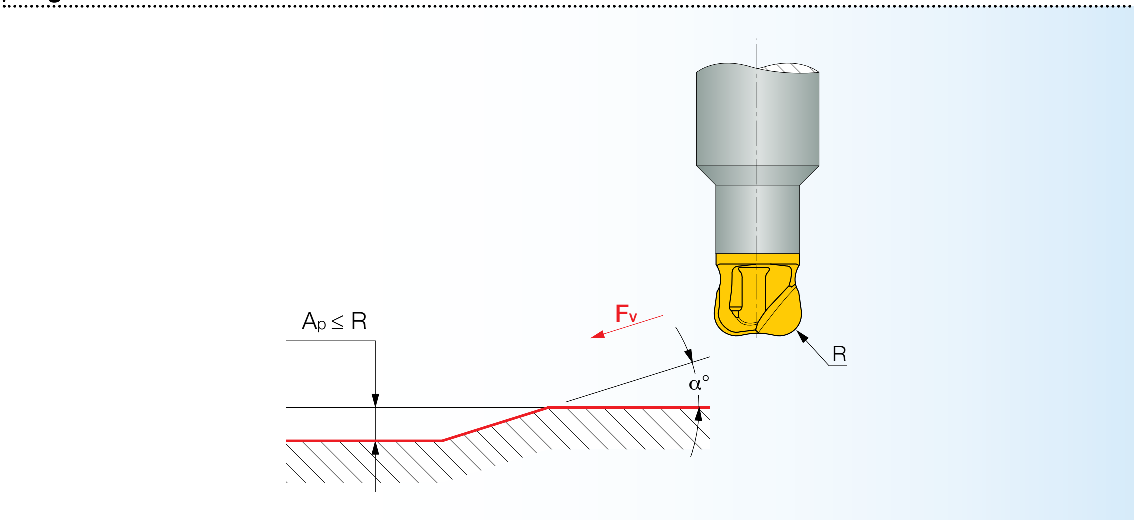
Example

In a die-making shop mill HCE D25-A-L170-C25 with insert HTR D250-R5.0-QF IC908 planned to be applied for rough milling of a mold insert cavity. The cavity material is stainless steel AISI 420F. The proposed cutting parameters are as below: the depth of cut – 2.5 mm, the width of cut – 17 mm. A rigid machine tool performs milling; and the cavity is properly secured in the workholding fixture of the machine tool. Find the programmed spindle speed and feed.

Basic feed for tooth fzo for groups 12-13 is equal to 0.14 mm/tooth (Table 53), chip thinning factor KTH for 25 mm dia. insert, 5 mm corner radius and 2.5 mm depth of cut (Table 54) – 1. Hence, the programmed feed will be 0.14 mm per tooth.

In accordance with Table 51 the recommended starting speed is 120 m/min. Effective diameter $De=25-2\times5+2\times\sqrt{(2\times2.5\times5-2.5^2)}=23.7$ (mm) that contains ~95% of the insert nominal diameter (25 mm). Therefore, the spindle speed can be calculated with the reference to the nominal diameter and it will be $1000\times120/(\pi\times25)=1528$ (rpm). Correspondingly, the feed speed becomes $0.14\times2\times1528=427.8$ (mm/min).

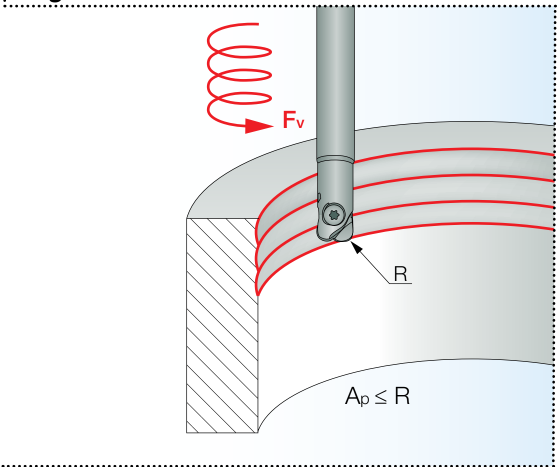
Fig. 20



Milling Tools

Die and Mold

Fig. 21



Toroidal cutting profile and rampdown milling

Toroidal milling tools can be used in rampdown milling applications whether linear ramping or ramping by helix (helical interpolation).
For each of these ramping techniques a depth of cut should be no more that the corner radius of a head, an insert or a solid tool (Fig. 20, 21).
Obviously, such is true also for milling by circular interpolation.

Marking the end of the chapter devoted to the toroidal tools, we can emphasize the following points:

- Toroidal tools are suitable for milling plane and contoured surfaces.
- Toroidal tools feature successful performance in face milling, helical and circular interpolation and plunging. Within certain limits they operate in linear ramping and peck drilling.
- Toroidal tools allow for substantial improvement in feed speed due to chip thinning effect and thus considerably reduces cycle time.
- The cutting profile of the toroidal tools has no point where the cutting speed will be zero during machining that gives convincing advantages - in comparison with ball nose cutters, for instance, and ensures smooth face milling with large width of cut.
- Such versatility and the sources of improving productivity determine the toroidal tools as exceptionally widespread in die and mold making for various applications from rough to finish milling, especially in machining cavities and faces.

Unexpectedly too large machining allowance

One of the most popular applications of toroidal tools is machining cavities by helical interpolation. A cavity may already be pre-shaped by casting or pre-machined. In this case the bottom of a cavity is connected with its walls by a fillet; and a machining allowance (material stock to be removed) for the filleted area increases. That may be problematic or even fatal for the tool (Fig. 22).
Therefore, in spite of impressive self-tuning possibilities of modern CNC machine tools with adaptive control always take the above into consideration and check thoroughly the machined shape and corresponding cutting data. In addition, the same picture may occur when machining outer surfaces (a punch, for instance) (Fig. 23).

Fig. 22

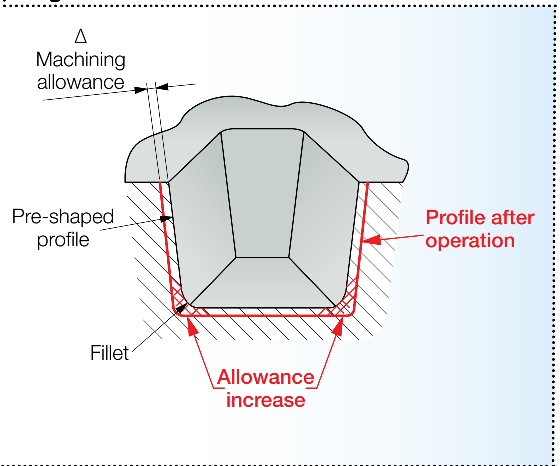
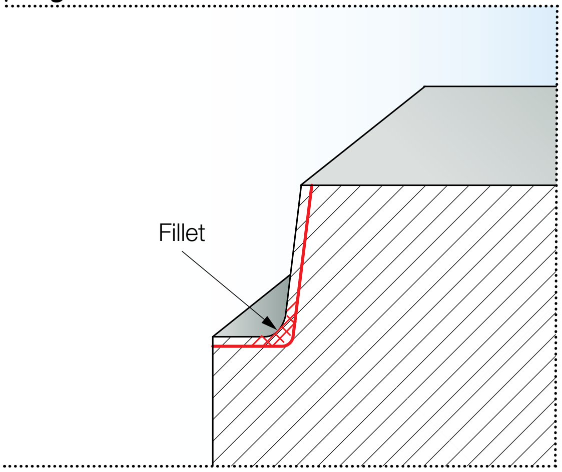


Fig. 23



3.2. Tools for High Feed Milling (HFM)

The notes regarding cutting edge angles in the section devoted to milling plane surfaces and especially the chip thinning phenomenon considered in the previous chapter are a good introduction to the main principle of high feed milling. As was shown, a shallow depth of cut combined with an appropriate cutting geometry allows for considerable increase in feed per tooth. In addition, such a combination minimizes the radial component of a cutting force and maximizes its axial component. Therefore, the resultant force of the components acts towards the spindle axis of a machine tool. As consequence, it causes substantial vibration reduction and correspondingly, stability of milling.

One would think, the toroidal tools can be a good solution for HFM. However, they have limited application for this method. First of all, even if a depth of cut remains be shallow, its slight increase may noticeably change the cutting edge angle, especially for the round inserts of small diameters or the toroidal milling heads, solid carbide tools or one-insert tools with small corner radii (Fig. 24). Further, an insert or a tooth produces a relatively narrow cut in such cases (W_1 and W_2 in Fig. 24). However, if the cutting edge will be an arc of a great cycle (Fig. 25), the limitations above will be overcome.

Fig. 24

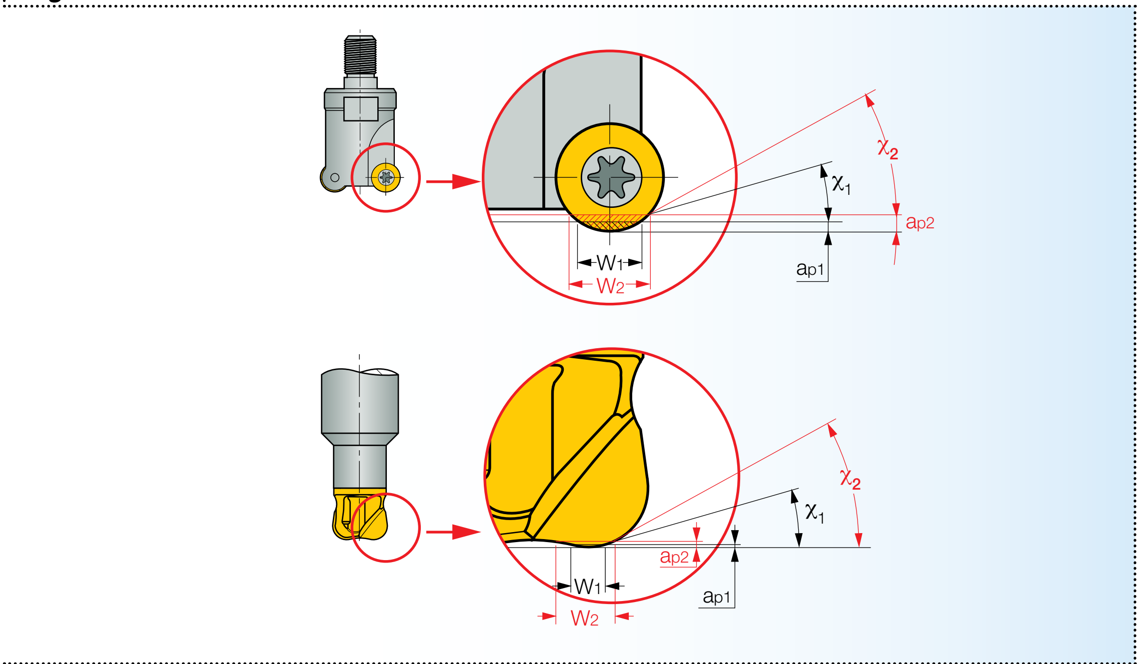
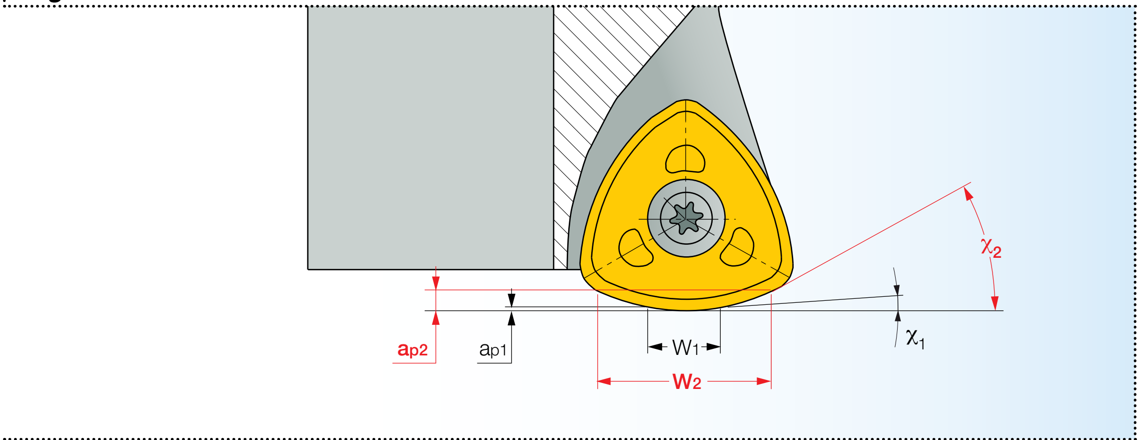


Fig. 25



Milling Tools

Die and Mold

In the past, the classical approach for rough milling dictated using as large-sized cutter as possible and required corresponding powerful machine tools. Today HFM leads to the same result that is reached by much smaller sized tools. No wonder many machine tools for HFM have spindles with a #40 taper, which can be enough for properly securing HFM tools and good operational performance.

Generally HFM relates to rough milling operations, but it can substantially reduce overall machining-cycle time. First of all, it allows for fast material removal, due to considerably increased feed rates. Second, the cutting force acting mainly towards to the spindle axis minimizes vibrations and enables stable rough milling at high overhangs – the property of the utmost importance for machining die cavities, punches and rams. Third, shallow depth of cut makes it possible to produce the contour that will be very close to the final shape and thus diminish or even eliminate semi finishing. All these features make HFM a significant time saver.

HFM is especially effective for machining die cavities or recesses by helical interpolation, principally if they have sufficiently large sizes. Naturally, HFM technique is successfully applied to rapid production of large-diameter holes directly from solid, or enlarging a previously made hole. In the case of a blind hole, the hole bottom will be flat. Moreover, the same HFM tool can produce holes of different diameters. In addition to the mentioned advantages, a HFM tool has far less contact with a cavity wall in contrast with a toroidal tool. In milling cavities by helical interpolation, the toroidal tool, working even at shallow depth of cut, comes in contact with the cavity material at increased area after some passes (Fig. 26), that caused vibrations and instable cutting.

However, the feed drive of a machine tool used for HFM should correspond to the required feed rates and be equipped by appropriate CNC devices. Further, rapid acceleration and slowing down combined with quickly changed working trajectories cause more intensive wear of the machine's moving parts such as guideways, feed screws, bearings, etc. Therefore, the concept of today's machining centers takes these factors into consideration; and machine tool builders produce the machine tools intended especially for HFM die and mold parts.

***Fig. 26:** 1st pass*

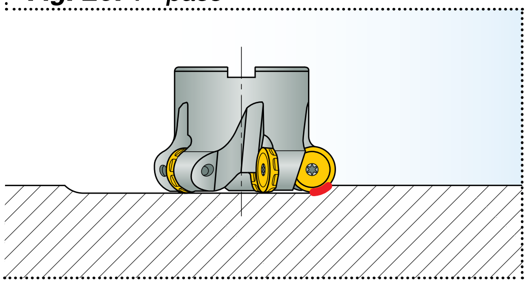


***Fig. 26:** 2nd pass*

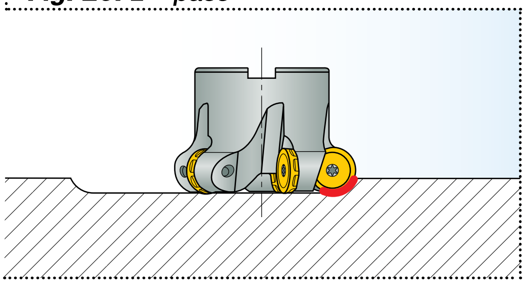


***Fig. 26:** 3rd pass*

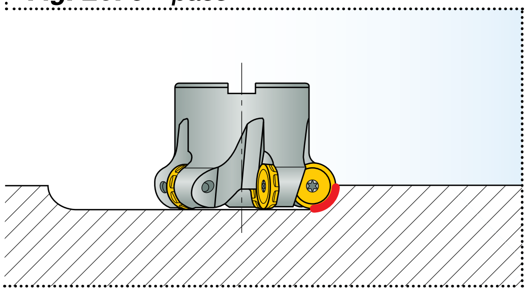
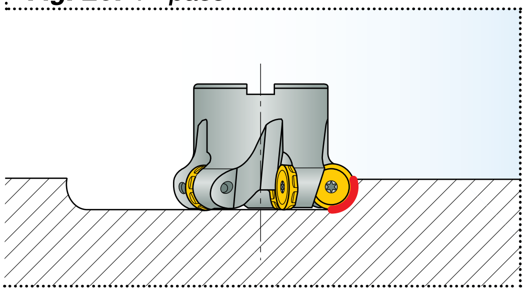


***Fig. 26:** 4th pass*



HFM and "old-type" CNC machine tools

Is the HFM method suitable for "old-type" CNC machine tools with spindle speeds normally 6000-7000 rpm? These machines are still common in many die and mold shops.
The answer is generally "yes" because the typical feed drive of the machine ensures necessary feed rates; and the mentioned values of the spindle speed are enough for rough milling of most die and mold materials. In HFM, the key for high material removal is incresed feed per tooth, combined with normal cutting speed at shallow depth of cut.

Originally the HFM tools were indexable milling cutters of relatively large nominal diameters. The replaceable inserts carried by the tools traditionally had trigon or quadrihedral shapes (Fig. 27). A cutting edge of the insert basically was an arc or two arc chords (correspondingly profiles 1 and 2 in Fig. 28). The progress in multiaxis CNC grinding and sharpening machines allowed for production of very complicated cutting geometries in tools of different, even small sizes. Eventually the HFM tools not only improved the shape of their inserts but were replenished by solid tools and interchangeable heads of less nominal diameters. Such a second birth of the HFM tools was very important for the die and mold industry; and since that moment the HFM technique is applied to roughing face surfaces or relatively large-sized shapes, as well as to various profiles of medium to small dimensions.

ISCAR's HFM line comprises families of indexable milling tools, solid carbide endmills and interchangeable **MULTI-MASTER** heads (the latter are intended for mounting in the **MULTI-MASTER** shanks)

Fig. 27

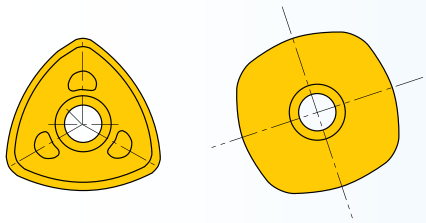


Fig. 28

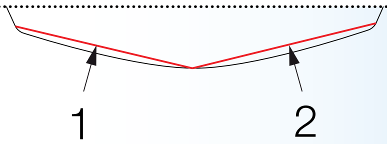


Fig. 29

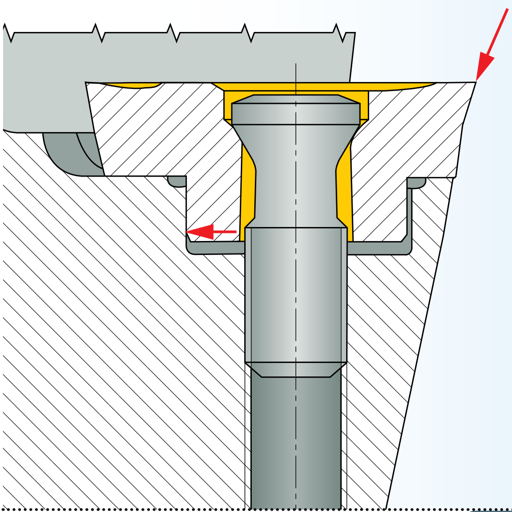


Fig. 30

Milling Tools

Die and Mold

3.2.1. Indexable Tools for HFM

FEEDMILL, the first-ever family of indexable tools, produced by ISCAR, feature the trigon inserts clamped in the tool body. The inserts have a cylindrical protrusion at their bottom, which when mounted into the corresponding hole of the insert pocket provides very rigid clamping, absorbing most of the forces usually exerted on the clamping screw (Fig. 29).

The quest for excellence of the HFM tools led to combining the **HELIDO**'s strength and **FEEDMILL**'s geometry, creating the **HELIDO UPFEED** family with the trigon double-sided inserts that feature 6 cutting edges (Fig. 30). The insert is secured into a dovetail inclined pocket, which provides very firm clamping. A cutting edge of the insert comprises two sections: the major (main or external) cutting edge and the minor (internal) cutting edge. This configuration improves a tool performance especially in rampdown milling, when the minor cutting edge plays a key role (Fig. 31). For better chip formation, the insert rake face portion adjacent to the major edge is convex; and the portion adjoining the minor edge – concave.

The mentioned advantages of the **HELIDO UPFEED** family allows effective rough milling at very high feeds for high metal removal. Tools of the family have a 17° cutting edge angle and are available in different configurations: shell mills, mills with shanks and milling heads for various adaptations.

Table 55 **Main Tool Families for HFM**

Family	Type	Tool configuration	Range of diameters, mm	No. of insert cutting edges	Tool (shank) designation	Insert (head) designation
HELIDO UPFEED	Indexable	Shell mills	40-125	6	FF FWX…	H600 WXCU…
		Shank-type mills	16-40		FF EWX…	
		FLEXFIT adaptation	20-40		FF EWX…M…	
		MULTI-MASTER adaptation	16-20		FF EWX…MMT…	
FEEDMILL	Indexable	Shell mills	40-125	3	FF FW…	FF WO…
		Shank-type mills	25-40		FF EW…	
		CLICKFIT adaptation	25-40		FF EW…CF…	
		FLEXFIT adaptation	25-40		FF EW…M…	
16FEEDMILL	Indexable	Shell mills	80	16	FF NM…	ONMU…
FEEDMILL	Solid	Solid carbide endmills	6-20	4*	EFF S4…	
MULTI-MASTER	Solid heads	Assembly: shanks with replaceable heads	10-25	2*	MM…	MM FF…
			8-25	4*		MM EFF…

* Number of flutes for solid carbide mills and **MULTI-MASTER** heads

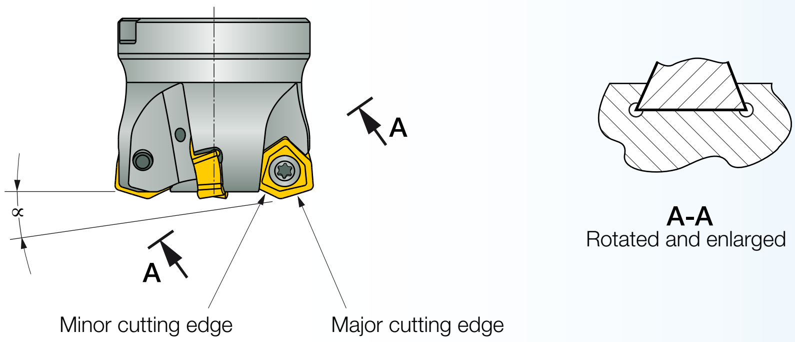
In accordance with the cutting geometry, there are two types of the **HELIDO UPFEED** inserts: T and HP. The T-type is designed for milling steel and cast iron, while the HP-type relates mostly to machining austenitic stainless steel and high temperature alloys. The T-type inserts have "I" marks on their top surfaces for visual identification (Fig. 30).

Referring to HFM, the die and mold industry is generally concerned with steel workpieces. Hence, the T-type inserts are more common there; however, the HP-type inserts are also used for manufacturing parts from corresponding materials.

Run faster

Two letters FF in the designation of ISCAR tools, inserts or heads for HFM means "Fast Feed". Run faster at fast feed for high productivity in rough milling operations!

Fig. 31



Starting Cutting Data

a) Depth of Cut Ap

Table 56 shows the depth of cut ranges, depending on the sizes of H600 WXCU…T inserts for **HELIDO UPFEED FF**… milling tools.

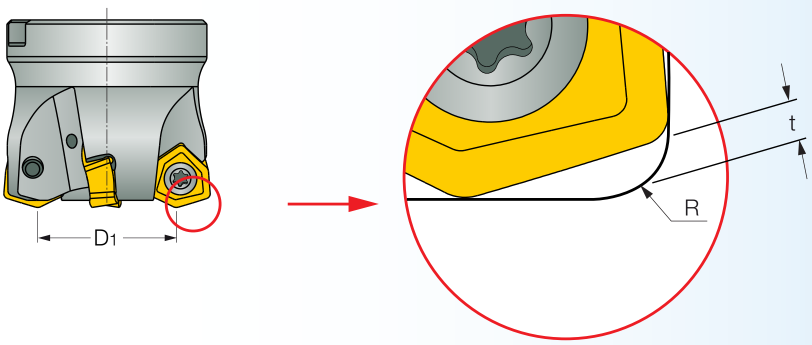
Table 56 **Depth of Cut Ap for FF… Tools with H600 WXCU…T Inserts**

Insert size	Insert designation	Ap, mm	
		min.	max.
04	H600 WXCU 04…T	0.2	0.8
05	H600 WXCU 05…T	0.25	1
08	H600 WXCU 08…T	0.4	2

Radius for programming

In CNC programming, an HFM tool is often specified as a milling cutter with a corner radius. The radius that is called as "radius for programming (R in Fig.32), relates to reference data and can be found in ISCAR catalogs and technical leaflets. The radius defines the maximal thickness of a cusp – a mismatch produced by such specification (correspondingly t in the same figure).

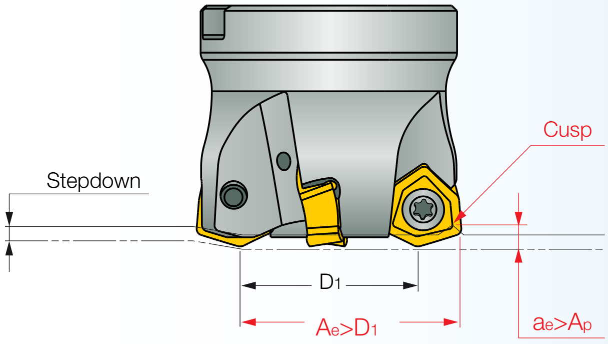
Fig. 32



b) Width of Cut Ae

It is strongly recommended that a width of cut be no more than diameter D1 in order to prevent tooth overloading, because of excess machining allowance in cusps produced on the further passes after stepdown (Fig. 33).

Fig. 33



c) Starting Feed Per Tooth fz

Normally, feeds per tooth for the majority of die and mold materials lay within the values shown in Table 57.
The values are enough for quick estimation of a feed per tooth and give satisfactory results when applied. More precisely, the feed fz can be defined with Table 58. For poor workholding, high overhang, milling near thin walls and in other cases of non-sufficient operational stability, the table data should be reduced by 30%.

Table 57 **Commonly Occurring Feeds for FF… Tools with H600 WXCU…T Inserts**

Insert size	Insert designation	Feed Fz, mm/tooth
04	H600 WXCU 04…T	0.4…1
05	H600 WXCU 05…T	0.5…1.5
08	H600 WXCU 08…T	0.6…2

Table 58 **Average Starting Feed fzo for FF… Tools with H600 WXCU…T Inserts**

ISO Class DIN/ISO 513	Workpiece material		Feed fzo, mm/tooth for FF…tools with H600 WXCU…T… Inserts**		
	Type	Material Group*	H600 WXCU 04	H600 WXCU 05	H600 WXCU 08
P	Plain carbon steel	1-4	1	1.3	1.7
		5	1	1.3	1.6
	Alloy steel and tool steel	6, 7	1	1.2	1.5
		8, 9	0.9	1.2	1.4
		10	0.8	1.1	1.3
		11	0.7	1	1.2
M	Martensitic s.s.	12, 13	0.9	1.2	1.4
K	Grey cast iron	15-16	1	1.3	1.6
	Nodular cast iron	17-18	0.8	1.1	1.3
H	Hardened steel	38.1	0.4	0.5	0.6
		38.2	0.3	0.3	0.4

* ISCAR material group in accordance with VDI 3323 standard
** In case of machining in unstable conditions, the starting feed should be reduced by 30%

Particular care: ramping down directly to solid

Starting high feed milling of cavities and pockets, an HFM tool performs the first cut by ramping down to a solid workpiece and then starts profile milling by helical interpolation. Keep in mind that in case, when the tool cuts downward, ramps to material, demands reduced feed per tooth; and the values shown in Table 58 should be decreased by 30-40%. Additionally, it is recommended to reduce the maximal depth of cut that is specified in Table 57 by 20%.

d) Starting cutting speed Vc

Table 59 comprises the averaged values of starting cutting speed Vc in relation to the insert carbide grades. If the cutting conditions are estimated as unfavorable (high overhang, poor work- or toolholding, etc.) the values should be reduced by 20-30%.

Table 59 **FF… Tools with H600 WXCU…T Inserts: Starting Cutting Speed Vc**

ISO Class DIN/ISO 513	Workpiece material		Vc, m/min for Grades			
	Type	Mat. group*	IC808	IC810	IC830	IC330
P	Plain carbon steel	1-4	150	150	150	135
		5	150	140	135	125
	Alloy steel and tool steel	6, 7	150	135	125	120
		8, 9	150	130	120	115
		10	130	125	115	110
		11	120	120	115	100
M	Martensitic s.s.	12, 13	120		120	120
K	Grey cast iron	15-16	200	220	220	
	Nodular cast iron	17-18	180	200	200	
H	Hardened steel	38.1	80	70		
		38.2	60	50		

* ISCAR material group in accordance with VDI 3323 standard

☐ – Recommended carbide grade

Depth of cut: work for maximum

The cutting edge angle of a FF tool with H600 inserts is practically consistant (17°) along the main cutting edge of an insert mounted in the tool. Therefore, reducing a depth of cut does not cause the chip thinning effect and does not allow for inscreasing feed per tooth and then run faster.
If your machine has enough power and workholding is rigid, work for maximal depth of cut – you gain cycle time and reach higher productivity.

Example

In order to cut cycle time for forging die manufacturing, a process planner decided to apply face mill tool FF FWX D080-06-32-08 with inserts H600 WXCU 080612T IC830 for rough milling a plane surface of a large-size block. The block material is AISI/SAE D2 tool steel, HB 190…210. The machining conditions can be estimated as stable: a machine tool with powerful main and feed drives, proper workholding, and short overhang of the tool.

Assuming that calculations show that the machine has enough power, the current manufacturing conditions allow the proposed maximal depth of cut (refer to remark "Depth of cut: work for maximum") 2 mm (Table 56).
The material relates to material group No.10. Starting feed fz=1.3 mm/tooth (Table 58), starting cutting speed Vc=115 m/min (Table 59).
Width of cut Ae=62 mm - refer to above paragraph b) "Width of cut Ae" and catalog data of the chosen tool (D1=64 mm).

Spindle speed N=1000×115/(π×80)=458 (rpm).
Feed speed V_F=1.3×6×458=3572 (mm/min).
Metal removal rate Q≈1.3×62×3572=287.9 (cm^3/min).

Particular care: milling full slot

If the HFM technique is applied for rough milling a full slot, starting feeds and cutting speeds recommended in Tables 58 and 59 should be reduced by 30%.

3.2.2. MULTI-MASTER milling heads and solid carbide endmills for HFM

There are two groups of the **MULTI-MASTER** heads for HFM: two-flute MM FF… heads of the "economy" type and multiflute MM EFF… heads with cutting geometry similar to the **FEEDMILL** solid carbide endmills. General data regarding these heads and solid endmills is shown in Tables 60 and 61.

Table 60 **Number of Flutes Z and Maximal Depth of Cut Apmax for MULTI-MASTER Heads and Solid Carbide Endmills Intended for HFM**

Type	Designation	Number of flutes Z	Apmax, mm, for nom. diam. D, mm						
			6	8	10	12	16	20	25
MULTI-MASTER	MM FF…	2			0.6	1	1.1	1.5	
MULTI-MASTER	MM EFF…	4		0.4	0.5	0.6	0.8	1	1.2
S.C. Endmills	EFF…	4	0.3	0.4	0.5	0.6	0.8	1	

Versatility of cutting tool (2) and …cutting insert

*ISCAR offers the customer inserts that when mounted into the existing tools, they turn these tools into HFM cutters. Inserts OFMW…FF that are intended for the **HELIOCTO** family and inserts ADKT… FF, ADCT … FF and APKT …FF, which are designed for the **HELIMILL** family, expand the application ranges of the mentioned lines for high-efficiency rough milling using the HFM method. The inserts do not require any new tools and are secured into pockets of the standard tools of the families. Further, face mill FF NM D080-06-27-R08 with 80 mm nominal diameter carries standard 16-edged insert ONMU 0806…. The insert orientation features similar capabilities to HFM tools. By using the tool in rough face milling, the material removal rate can be increased by 1.5 to 2 times, when compared with the standard tool with this insert. By the way, due to its 16 cutting edge insert ONMU 0806…is a winning economical solution for high feed milling of large-sized plane surfaces. The ISCAR main catalog and technical leaflets contain necessary information regarding these solutions based on the principle of versatility, where the right combination of indexable tools and inserts result in new positive results.*

Table 61 **Maximal Width of Cut Aemax* for MULTI-MASTER Heads and Solid Carbide Endmills Intended for HFM**

Designation	Max. Aemax, mm, for nom. diam.D, mm						
	6	8	10	12	16	20	25
MM FF... heads			7.7	9.1	13.6	17.5	
MM EFF... heads and EFF... endmills	4.5	6.1	7.7	9.1	12.5	15.5	19.5

* With the assumption that the depth of cut is Apmax, shown in Table 60

A more accurate estimatwe for depth and width of cut with respect to a workpiece material can be obtained from Tables 62-64. The tables contain average values that work well when applied to defining starting cutting data.

Table 62 **MULTI-MASTER Heads MM FF...: Average Depth of Cut Ap, mm**

ISO Class DIN/ISO 513	Material Group*1	Relation	Ap for D, mm			
			10	12	16	20
P	1-9	~0.06 D	0.6	0.7	1	1.2
P	10-11	0.05 D	0.5	0.6	0.8	1
M	12, 13	0.05 D	0.5	0.8	0.8	1
K	15-18	Apmax	0.6	1	1.1	1.4
H	38.1*2	~0.045 D	0.4	0.5	0.7	0.9
H	38.2*3	~0.03 D	0.3	0.4	0.5	0.6
H	39*4	~0.02 D	0.2	0.25	0.3	0.4

*1 ISCAR material group in accordance with VDI 3323 standard
*2 HRC 45-49
*3 HRC 50-55
*4 HRC 56-63

Table 63 **S.C. Endmills EFF... and MULTI-MASTER Heads MM EFF...: Average Depth of Cut Ap, mm**

ISO Class DIN/ISO 513	Material Group*1	Relation	Ap for D, mm						
			6	8	10	12	16	20	25
P	1-9	~0.045 D	0.3	0.35	0.45	0.55	0.75	0.9	1.1
P	10-11	~0.04 D	0.25	0.3	0.4	0.5	0.65	0.8	1
M	12, 13	~0.04 D	0.25	0.3	0.4	0.5	0.65	0.8	1
K	15-18	Apmax	0.3	0.4	0.5	0.6	0.8	1	1.2
H	38.1*2	~0.035 D	0.2	0.25	0.35	0.45	0.6	0.7	0.9
H	38.2*3	~0.03 D	0.2	0.25	0.3	0.4	0.5	0.6	0.75
H	39*4	~0.02 D	0.1	0.15	0.2	0.25	0.3	0.4	0.5

*1 ISCAR material group in accordance with VDI 3323 standard
*2 HRC 45-49
*3 HRC 50-55
*4 HRC 56-63

Particular care: milling full slot

*Speaking about application of the solid carbide endmills and the **MULTI-MASTER** heads intended for HFM, it should be noted that all additional limitations regarding table values of starting cutting data considered for **HELIDO UPFEED** cutters are also valid here:*

- *Reduce feed by 30% in case of poor stability*
- *Decrease feed by 30-40% and maximal depth of cut by 20% for ramping down to solid*
- *Reduce feed and cutting speed by 30% when milling full slot*

Table 64 **MULTI-MASTER Heads MM FF.../MM EFF... and S.C. Endmills EFF...: Average Width of Cut Ae, mm**[*1]

ISO Class DIN/ISO 513	Material Group[*2]	Relation	Ae for D						
			6	8	10	12	16	20	25
P	1-9	~0.7 D	4	5.5	7	8.5	11.5	14	17.5
	10-11	~0.6 D	3.5	4.5	6	7	9.5	12	15
M	12, 13	~0.6 D	3.5	4.5	6	7	9.5	12	15
K	15-18	~0.7 D	4	5.5	7	8.5	11.5	14	17.5
H	38.1[*3]	~0.45 D	2.5	3.5	4.5	5.5	7	9	11
	38.2[*4]	~0.3 D	2	2.5	3	3.5	4.5	6	7.5
	39[*5]	~0.25 D	1.5	2	2.5	3	4	5	6

*1 With reference to depth of cut, Ap shown in Table 63
*2 ISCAR material group in accordance with VDI 3323 standard
*3 HRC 45-49
*4 HRC 50-55
*5 HRC 56-63

Table 65 **MULTI-MASTER Heads MM FF.../MM EFF... and S.C. Endmills EFF...: Starting Speed Vc, m/min**

ISO Class DIN/ISO 513	Material Group*	Vc, m/min
P	1	180
	2-4	160
	5-6	150
	7-9	140
	10	130
	11	120
M	12, 13	120
K	15-16	180
	17-18	160
H	38.1	100
	38.2	80
	39**	60

* ISCAR material group in accordance with VDI 3323 standard

Table 66 **MULTI-MASTER Heads MM FF...: Starting Feed fzo, mm/tooth**

ISO Class DIN/ISO 513	Material Group[*1]	Relation	Ap for D, mm			
			10	12	16	20
P	1-9	~0.06 D	0.6	0.7	1	1.2
	10-11	0.05 D	0.5	0.6	0.8	1
M	12, 13	0.05 D	0.5	0.8	0.8	1
K	15-18	Apmax	0.6	1	1.1	1.4
H	38.1[*2]	~0.045 D	0.4	0.5	0.7	0.9
	38.2[*3]	~0.03 D	0.3	0.4	0.5	0.6
	39[*4]	~0.02 D	0.2	0.25	0.3	0.4

*1 ISCAR material group in accordance with VDI 3323 standard
*2 HRC 45-49
*3 HRC 50-55
*4 HRC 56-63

Tables 65-67 show recommended values for starting cutting speed and feed per tooth.

Example
In a mold making shop there is a part to be machined by four-flute solid carbide endmill EFF-S4-12 045/34C12R2.0M. The part material is AISI P20 mold steel, HRC 32…35. The operational stability is good enough.
Find starting cutting data.

The tool diameter is 12 mm. The material relates to material group No.6. Define the depth of cut as 0.55 mm (Table 63) and the width of cut as 8.5 mm (Table 64).
The starting cutting speed is 150 m/min (Table 65) and the starting feed is 0.48 mm/tooth (Table 67).

Calculations:
- Spindle speed 1000×150/(π×12)=3980 (rpm)
- Feed speed 0.48×4×3980=7641.6 (mm/min)
- Approximately metal removal rate 0.55×8.5×7641.6/1000=35.7 (cm^3/min)

Table 67 **S.C. Endmills EFF… and MULTI-MASTER Heads MM EFF…: Starting Feed fzo, mm/tooth**

ISO Class DIN/ISO 513	Material Group[*1]	fzo, for D, mm						
		6	8	10	12	16	20	25
P	1-4	0.35	0.48	0.57	0.67	0.75	0.9	1
	5	0.33	0.43	0.5	0.57	0.65	0.75	0.87
	6, 7	0.28	0.33	0.4	0.48	0.57	0.67	0.78
	8, 9	0.25	0.3	0.35	0.43	0.52	0.6	0.7
	10	0.22	0.28	0.33	0.38	0.48	0.57	0.67
	11	0.2	0.25	0.3	0.35	0.43	0.52	0.62
M	12, 13	0.25	0.3	0.35	0.43	0.52	0.6	0.7
K	15-16	0.34	0.45	0.52	0.6	0.7	0.8	0.9
	17-18	0.3	0.38	0.45	0.52	0.6	0.7	0.8
H	38.1[*2]	0.16	0.2	0.25	0.33	0.4	0.48	0.55
	38.2[*3]	0.12	0.16	0.22	0.3	0.38	0.45	0.52
	39[*4]	0.1	0.12	0.16	0.16	0.2	0.2	0.25

*1 ISCAR material group in accordance with VDI 3323 standard
*2 HRC 45-49
*3 HRC 50-55
*4 HRC 56-63

Depth of cut and stepover (pick feed)

In HFM with stepover passes, the depth of cut and the stepover are interrelated quantities. Because the cutting edge of a EFF solid carbide endmill or a MM FF/EFF milling head actually is an arc of large-diameter circle, the maximal stepover is limited by chord length t defined by depth of cut ap (Fig. 34). Therefore, the width of cut in Tables 61 and 64 specifies the values that are not recommended to exceed when the depth of cut is defined by corresponding data in Tables 60, 62 and 63. If not, the produced cusp shape can lead to the tool (head) overloading in further milling with stepdown. It is clear that decreasing the stepover reduces the cusps (Fig. 35) and improve surface condition.

Generally, in machining large-sized, mostly plane, surfaces with an allowance no more than the recommended depth of cut, HFM with maximal corresponding width of cut will be the most productive. In case when the allowance is divided between stepdown passes, the stepover definition requires more careful examination, in order to avoid overload after stepdown.

Fig. 34

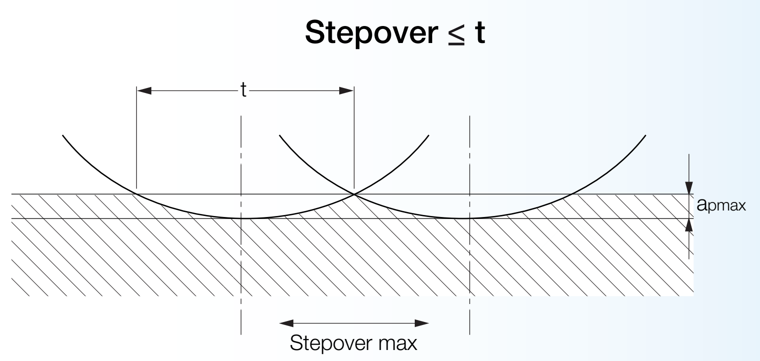
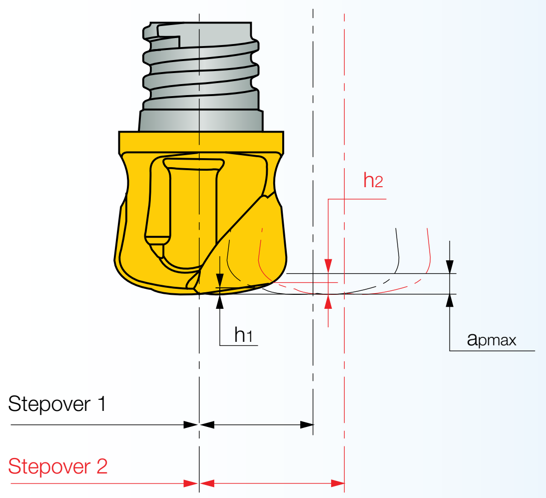


Fig. 35



3.3. Ball nose milling tools

A ball nose milling tool has a ball-shaped (spherical) cutting edge. This means that the cutting edge is a part of a sphere (Fig. 36). Nothing other than a ball has the common normal to a surface; and actually only a spherical cutting tool can ensure theoretical pinpoint contact with a required 3-D surface. Thus the tool does not cause deformation of the needed shape while milling (Fig. 37).

Ball mill

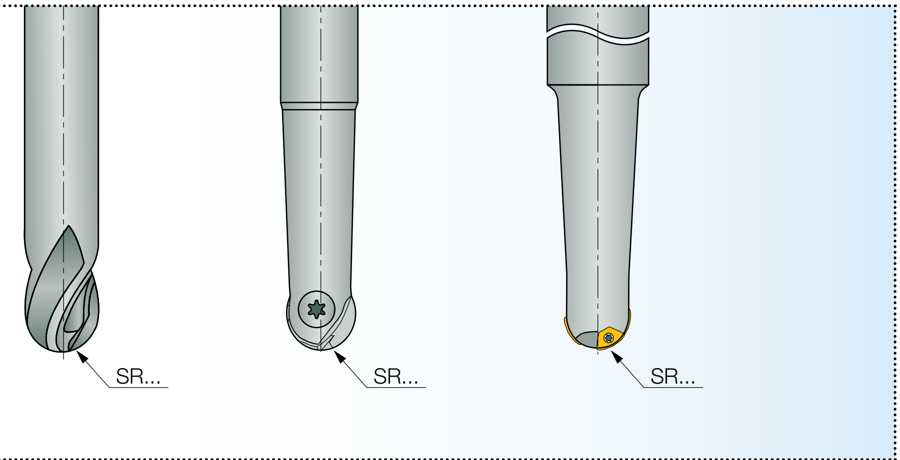
From time to time (especially in shoptalk of the die and mold professional environment) the ball nose cutters are said to be "ball mills". Such a definition should be avoided because ball mills refer to grinding devices of the specific design principle and relate to grinding materials into powder, but not to material removal by milling.

The die and mold industry feature no lack of complex shaped parts - the ball nose milling tools excellently meet the requirements for machining these parts. It is little wonder when speaking about specific features of the cutting tools in die and mold making, the reference is to the ball nose and the toroidal cutting profiles.

In the ball nose cutters, one of the important engineering factors is the angular value of a ball-shaped cutting edge. Usually it is 180° (hemisphere, Fig. 38); but tools with the edge more than hemisphere (bulb-type cutters, Fig. 39), typically 220°-250°, and the edge less than hemisphere (taper or tapered ball nose cutters, Fig. 40), basically 80°-89°, are common in die and mold making.

The ball nose milling tools with the ball-shaped cutting edge as a hemisphere often have also a cylindrical cutting portion. The portion allows for better surface finish, while machining near straight walls and enables for increasing step down for milling passes (compare cases a and b in Fig. 41).

Fig. 36



Milling Tools

Die and Mold

Fig. 37

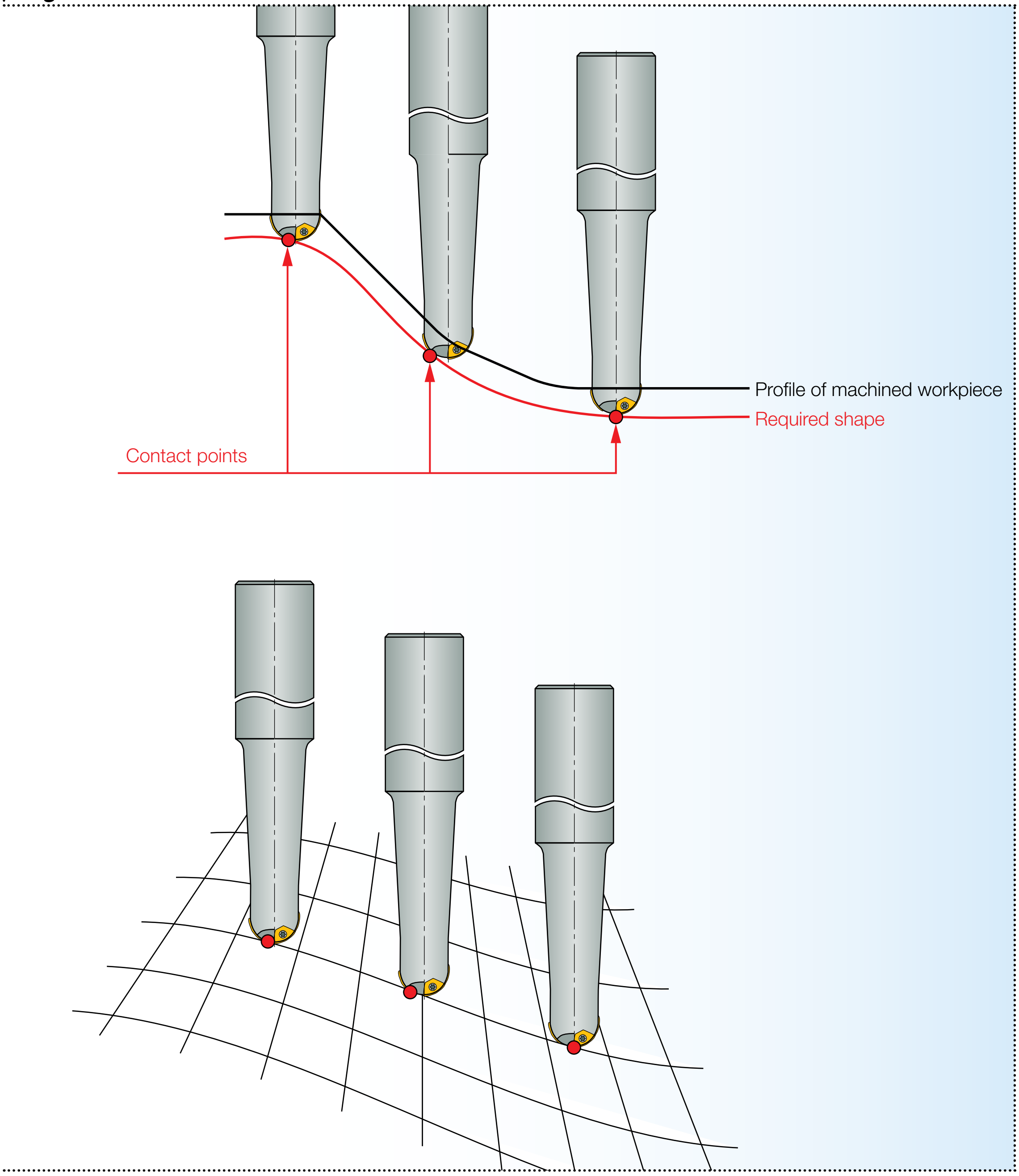

Profile of machined workpiece
Required shape
Contact points

Fig. 38

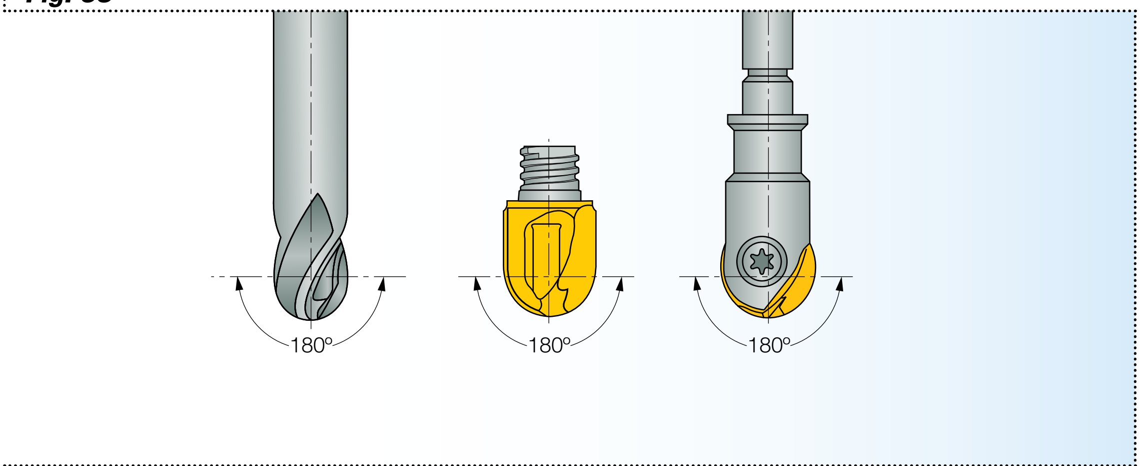

180º
180º
180º

Fig. 39

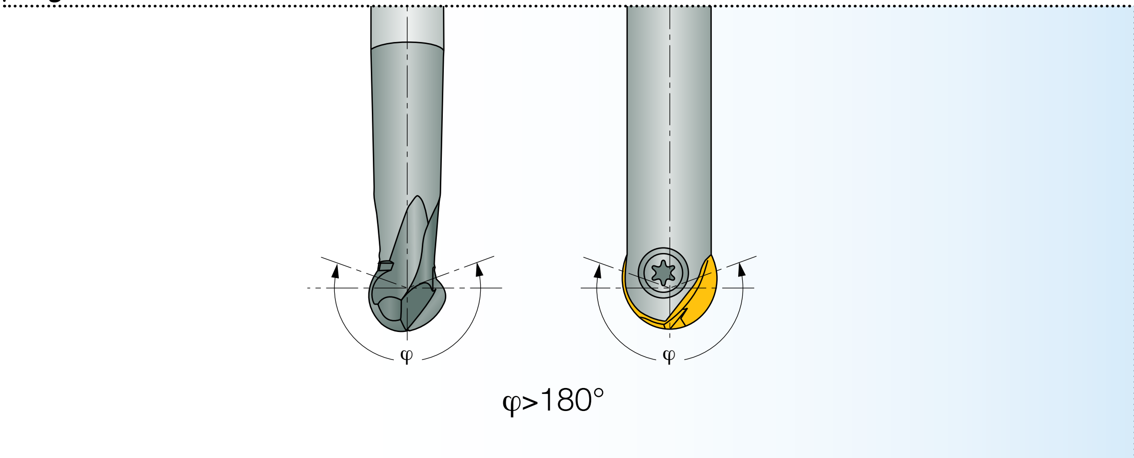


Fig. 40

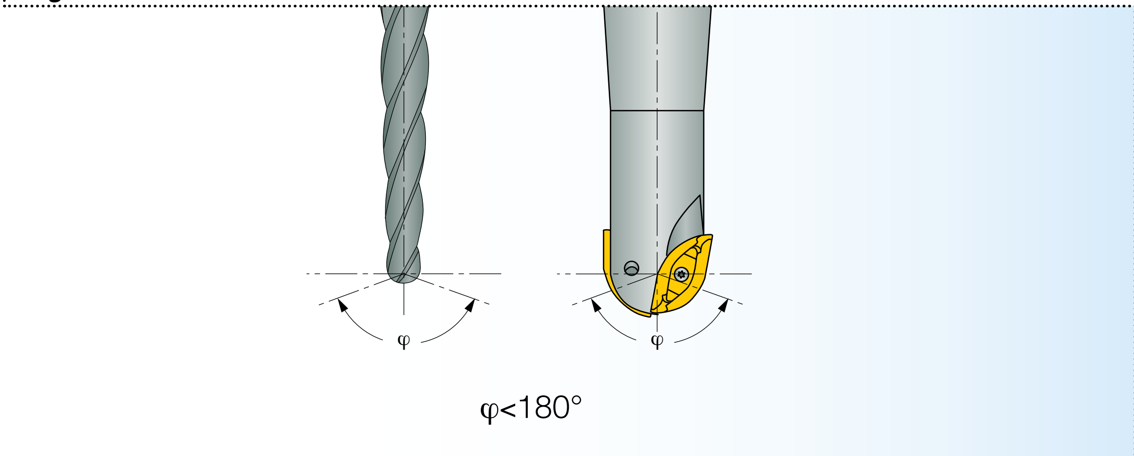


Fig. 41 a

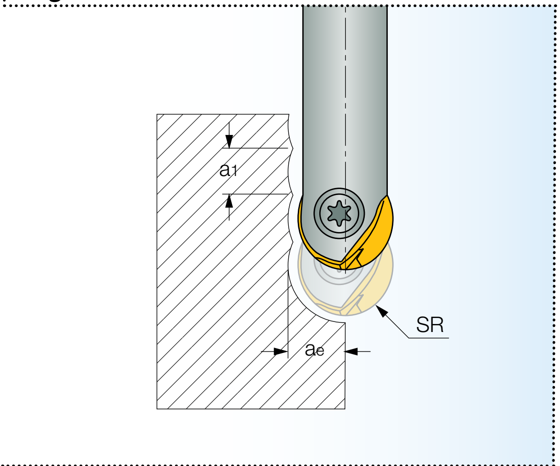
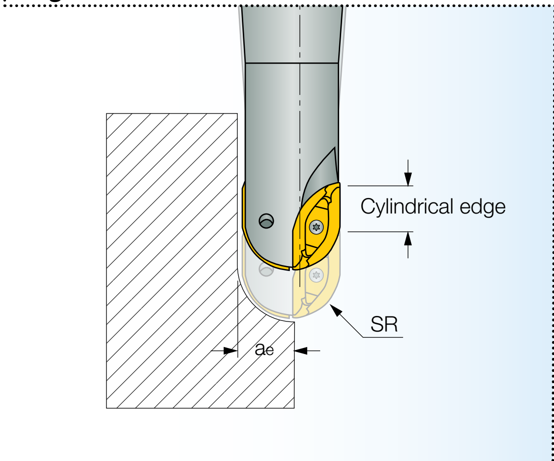


Fig. 41 b



The ability to generate exact, true to form surfaces is the prime advantage of the ball nose milling tools. However, the ball-shaped cutting edge, which predetermines this important feature, has one serious weak point: the zero velocity (and hence the zero cutting speed) of a cutter tip. That phenomenon makes cutting near to the tip difficult. Further, the points of a ball-shaped cutting edge lay on unequal distance from a tool axis varying from zero (the tip) to the radius of the corresponding sphere. Such a variation means that the points cut were with different cutting speeds. The chip thinning effect considered in the previous sections of the guide also takes place here. The combination of unequal cutting speeds and dissimilar chip thickness leads to a substantial difference in loading the points of the cutting edge along its profile, which makes cutting harder and intensifies wear in the certain areas of the cutting edge.

Cutting tool engineers take into account the mentioned negative effect when designing the cutting geometry of a ball nose tool. Additionally, machining practice advances different methods that improve the performance of the ball nose tools and makes them more effective. For instance, milling with a tool when its axis is not perpendicular to a machined surface (“tilting”, Fig. 42) makes loading on the cutting edge more uniform. Another example: replacing rampdown milling (case a, Fig. 43) on rampup milling (case b, the same figure) in a machining process at once changes the cutting conditions for the better. Under the same programmed spindle speed and feed in case a), the most stressed portion of the cutting edge is the area near the cutter tip that features low cutting speeds; while in case b), the cutting edge area, which carries the main load, is in a much better situation when a more acceptable cutting speed corresponds to this area. Of course, machining specific parts can demand various milling strategies; and the examples only illustrate some particular properties associated with the ball nose cutting geometry. The correct process planning requires taking the properties into consideration.

ISCAR carries a wide range of ball nose milling cutters of diverse types: indexable and solid, single-insert and with interchangeable solid cutting heads. Normally these are endmills with shank and they vary in dimensions and obtainable accuracy. In addition, the cutters of the single-insert and indexable types are available not only as tools with integral body but in most cases as replaceable cutting heads with a **FLEXFIT** or **MULTI-MASTER** adaptation. For the tools with integral body (Fig.44), there are different design configurations with straight (type A) and tapered (conical) neck (types B and D). Usually type B features an operating angle α equal to 5°, and type D –2°. Table 68 shows the most popular ISCAR families of the ball nose endmill cutters.

Historical background: copy mill

In many cases the ball nose milling tools are called “copy milling cutters”, “copying style mills” or plainly “copy mills”. These terms trace their history to not so long-ago when the contoured surfaces were produced on conventional milling machines of a copying arrangement – the copy milling machines. The machines, operated manually or having a follow-up drive of hydraulic, electric, etc. type, feature the ability to follow a master model (template) and thus to generate the contoured surfaces by cutting. As a matter of course, the copy milling machines use mainly the cutters of ball-shaped or toroidal cutting profile that were called “copy mills” accordingly.
Introduction of the CNC machines dramatically changed the technology of generating the contoured surfaces by cutting. In contrast to the traditional copy milling that allowed machining for the most part 2-D profiles, modern CNC technology is capable of producing very complicated 3-D shapes by metal cutting; and today “template”, which defines tool paths, is a computer solid model.
The model built with the use of CAD/CAM software is intended for generating corresponding CNC programs. Therefore sometimes used terms “copy milling” and correspondingly “copy mills”, now substantially differ from their original meanings.

Fig. 42

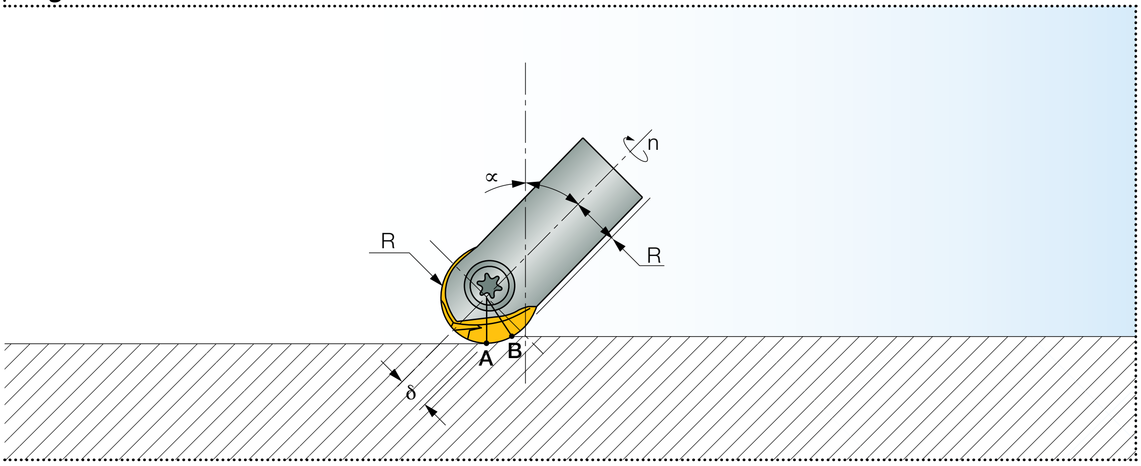


Fig. 43 a

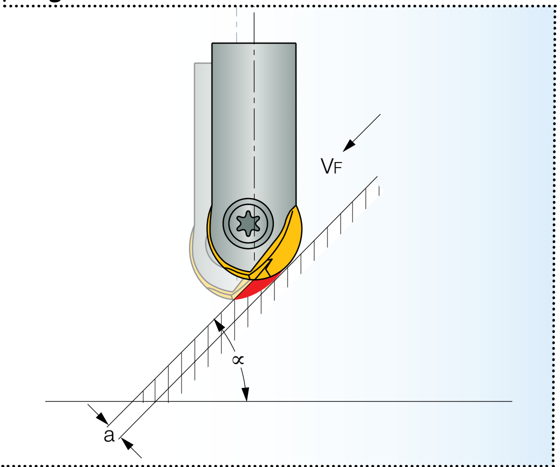


Fig. 43 b

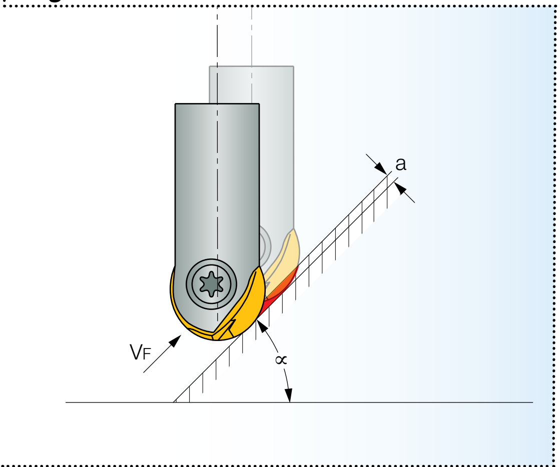
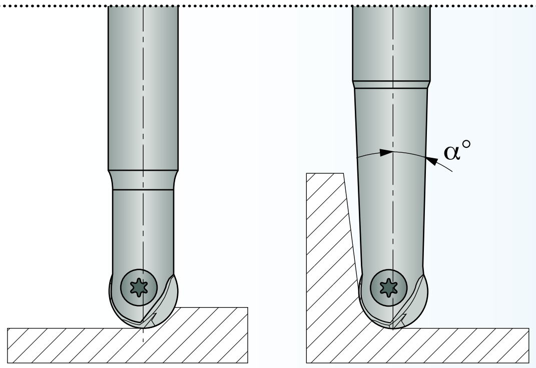


Fig. 44



Type A

Type B and D

Table 68 **Ball Nose Endmill Cutters: Main Families**

Family	Type	Diameter Range, mm*
BALLPLUS	Single-insert	12…25
DROPMILL	Indexable	12…50
HELIBALL	Single-insert	8…10
MULTI-MASTER	Interchangeable solid heads	6…25
SOLIDMILL	Solid carbide tools	0.4…25

* For production of the standard line

Tilting action: case study

When milling with a ball nose cutter, tilting the cutter axis prevents the non-cutting effect of the cutter tip due to its zero cutting speed.
Turn to Fig. 42. Section AB of the cutting edge performs milling operation, removing machining allowance δ.
The cutting speed within the section is directly proportional to a distance from a point of the section to the cutter axis and it is minimal in point A. The cutting speed has a maximum in the point spaced on distance R from the axis, where R – the radius of the cutter sphere. The distance from point A to the axis can be found as $R \cdot \sin\alpha$, where α – a tilting angle.
It is not difficult to see that if the angle is 5°, the cutting speed in point A is approximately 9% of the maximum, if 10° - 17%; and for the angle 15° the speed is already 26% of the maximum.

Starting Cutting Data and Tool Selection

As already stated, the cutting speed varies along the section of a ball-shaped cutting edge that engages directly in milling. Due to the chip thinning effect the chip thickness also varies within the section. A correctly selected milling method demands removing most of a machining allowance by the area with the possibly maximal cutting speed. Compare, for example, the mentioned cases a) and b) in Fig. 43. The latter is the correct choice, while the first case shows cutting in unfavorable conditions and should be avoided.

For a ball nose milling tool the effective cutting diameter (see the marginal note on page 58) De, which relates to the point of the cutting edge with maximal cutting speed, can be defined from the following equation:

$$De = 2 \times \sqrt{(D \times ap - ap^2)} \quad (15)$$

Where D – tool diameter
ap – axial depth of cut

However, the equation will not work for ramping and milling near straight walls (cases b) and c) in Fig. 45) where the areas of the cutting edge having diameters more than De determined from equation 15. Table 69 shows how to define the effective diameter, depending on a cutting path.

Fig. 45 a

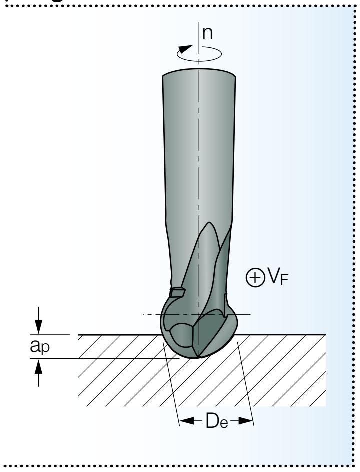


Fig. 45 b

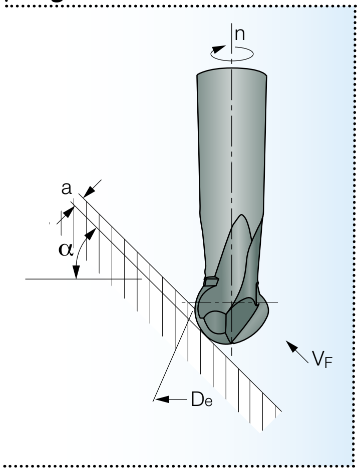


Fig. 45 c

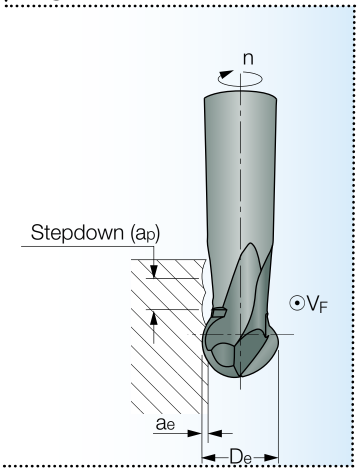


Table 69 **Effective Diameter for Ball Nose Milling Cutters**

Cutting Path	Case in Fig. 45	Effective Diameter De	Notes
The cutter axis is normal to a machined surface	a)	$2\times\sqrt{(D\times a_p - a_p^2)}$	
Ramping, milling inclined surfaces	b)	$(D-2\times a)\times\sin\alpha+2\times\sqrt{(D\times a-a^2)}\times\cos\alpha$*	$\approx D\times\sin\alpha$**
Milling straight walls	c)	D	

* α - 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 source of serious inaccuracy.

Be accurate with effective diameter

In milling with ball nose cutters, especially in ramping operations, it is very important to be accurate in computation of the effective diameter. Precise calculation leads to correct cutting action and better performance.

Example

Calculate the effective diameter for a ball nose cutter of 10 mm diameter, which is applied to ramp up milling with a ramping angle of 30°. The machining allowance 0.4 mm should be removed by one pass.

In accordance with Table 69 (case b) in Fig. 45):
$De=(10-2\times0.4)\times\sin30°+2\times\sqrt{(10\times0.4-0.4^2)}\times\cos30°=7.99$ (mm) ≈8 mm.
By comparison, the simplified equation will give $10\times\sin30°=5$ (mm) - the truncation error is 37.5%.

If for the same cutter the ramping angle will be 10° and the allowance 2 mm, calculated effective diameter $De=(10-2\times2)\times\sin10°+2\times\sqrt{(10\times2-2^2)}\times\cos10°=8.9$ (mm), while the simplified equation gives only 1.7 mm! The truncation error in this case reaches 80%!

On the contrary, for 70° ramping with a 25 mm diameter ball nose cutter with a 1 mm allowance $De=(25-2\times1)\times\sin70°+2\times\sqrt{(25-1)}\times\cos70°=24.96$ (mm) ≈25 mm; and in accordance with the simplified equation - $25\times\sin70°=23.5$ (mm).
The truncation error here is only 6% and that is totally acceptable.
One can easily see that equation (15) is nothing but a particular case of the formula for the effective diameter of a ball nose milling cutter in ramping applications and is simply obtained from the latter by substitution zero value for angle α.

The ball-shaped (spheric) cutting edge is the cause of chip thinning (Fig. 46). Just as the milling tools with toroidal cutting profile, for a ball nose cutter the variation of the depth of cut changes the maximal cutting angle. The chip thinning effect has been already considered in reasonable detail in the section related to the toroidal tools; and it is not necessary to repeat it again. However it is very important to re-emphasize that the chip thinning factor increases the programmed feed per tooth in order to obtain the planned maximal chip thickness, shall be taken into account. The chip thinning factor is a function of the cutter spheric diameter D and the axial depth of cut ap (Fig. 47).

Normally, chip thinning factor KTH can be found from the following equation:

$$KTH = 1/\sin \chi_{max} \qquad (16)$$

Maximal cutting edge angle χmax, as it follows from equation (4), is determined by

$$\chi_{max} = \arccos\,(1-(2 \times ap/D)) \qquad (4a)$$

Example

What feed and spindle speed should be programmed for a milling operation performed by 16 mm diameter two-flute ball nose cutter, if the required cutting speed is 100 m/min and the expected maximal chip thickness 0.12 mm? The axial depth of cut is 3.5 mm.

Effective diameter $De=2\times\sqrt{(16\times3.5-3.5^2)}= 13.2$ (mm)
Correspondingly, programmed spindle $n=1000\times100/(\pi\times13.2)= 2411$ (rpm)
Maximal cutting edge angle $\chi max = \arccos\,(1-(2\times3.5/16))= 55.8°$
Chip thinning factor $KTH = 1/\sin \chi max=1/\sin55.8°= 1.2$
Thus feed per tooth $fz=0.12\times1.2= 0.14$ (mm/tooth) and programmed feed speed
$V_F=0.14\times2\times2411= 675$ (mm/min)

As a comparison, ignoring such factors as effective diameter and chip thinning results in spindle speed $n=1000\times100/(\pi\times16)= 1990$ (rpm) and programmed feed speed $V_F= 0.12\times2\times1990= 477.6$ (mm/min). Under this programmed data, productivity of the operation will be approximately 30% less!

Fig. 46

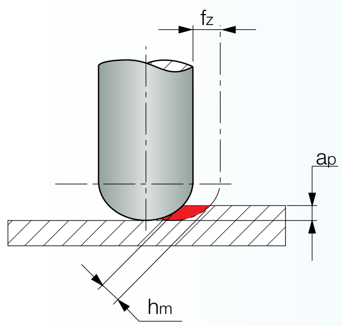
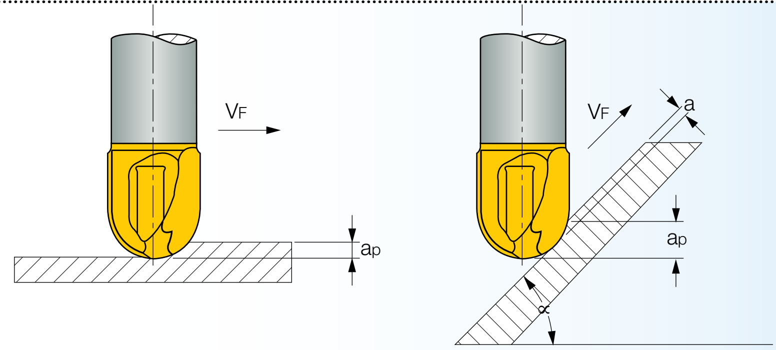


Fig. 47



In addition to equations (15) and (16), effective cutting diameter De and chip thinning factor KTH can also be found from Table 70. Just as well, the table data is very useful for quickly estimating cutting data.

In milling with ball nose cutters, correct feed and speed calculations should be based on the corresponding effective diameter of a cutter and the chip thinning factor defined by application. Leaving them out can lead to to poor operational performance and less productivity.

Table 70 **Effective Cutting Diameter De, mm, and Chip Thinning Factor KTH for Ball Nose Milling Cutters**

ap, mm	**D, mm**															
	SØ4		SØ5		SØ6		SØ7		SØ8		SØ10		SØ12		SØ16	
	De	KTH	De	KTH	De	KTH	De	KTH	De	KTH	De	KTH	De	KTH	De	KTH
0.2	1.7	2.3	1.9	2.5	2.1	2.8	2.3	3	2.5	3.2	2.8	3.6	3.1	3.9	3.5	4.5
0.3	2.1	1.9	2.4	2.1	2.6	2.3	2.8	2.5	3	2.6	3.4	2.9	3.7	3.2	4.3	3.7
0.5	2.6	1.5	3	1.7	3.3	1.8	3.6	1.9	3.9	2.1	4.3	2.3	4.8	2.5	5.6	2.9
0.7	3	1.3	3.5	1.4	3.8	1.5	4.2	1.7	4.5	1.8	5.1	1.9	5.6	2.1	6.5	2.4
1	3.5	1.1	4	1.2	4.5	1.3	4.9	1.4	5.3	1.5	6	1.7	6.6	1.8	7.7	2.1
2	4	1	4.9	1	5.6	1.1	6.3	1.1	6.9	1.1	8	1.2	8.9	1.3	10.6	1.5
3	—	—	—	—	6	1	6.9	1	7.7	1	9.1	1.1	10.4	1.1	12.5	1.3
4	—	—	—	—	—	—	—	—	8	1	9.8	1	11.3	1.1	13.8	1.1
5	—	—	—	—	—	—	—	—	—	—	10	1	11.8	1	14.8	1.1
6	—	—	—	—	—	—	—	—	—	—	—	—	12	1	15.5	1
7	—	—	—	—	—	—	—	—	—	—	—	—	—	—	15.9	1
8	-	—	—	—	—	—	—	—	—	—	—	—	—	—	16	1

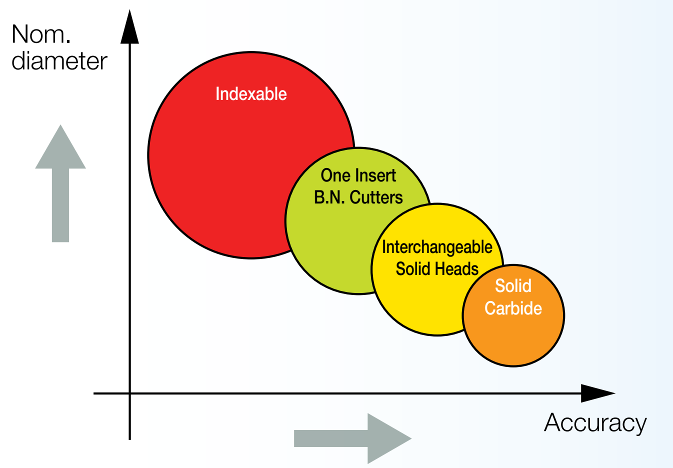
Table 70 (cont.) **Effective Cutting Diameter De, mm, and Chip Thinning Factor KTH for Ball Nose Milling Cutters**

ap, mm	D, mm									
	SØ20		SØ25		SØ32		SØ40		SØ50	
	De	KTH	De	KTH	De	KTH	De	KTH	De	KTH
0.3	4.9	4.1	5.4	4.6	6.2	5.2	6.9	5.8	7.7	6.5
0.5	6.2	3.2	7	3.6	7.9	4	8.9	4.5	9.9	5
1	8.7	2.3	9.8	2.5	11.1	2.9	12.5	3.2	14	3.6
3	14.3	1.4	16.2	1.5	18.6	1.7	21	1.9	23.7	2.1
5	17.3	1.1	20	1.2	23.2	1.4	26.4	1.5	30	1.7
8	19.6	1	23.3	1.1	27.7	1.1	32	1.2	36.7	1.4
10	20	1	24.5	1	29.7	1.1	34.6	1.1	40	1.2
12	—	—	25	1	30.1	1	36.7	1.1	42.7	1.2
16	—	—	—	—	32	1	39.2	1	46.6	1.1
20	—	—	—	—	—	—	40	1	49	1
25	-	—	—	—	—	—	—	—	50	1

Apart from everything else, defining cutting data depends on the type of the ball nose cutters. Normally, the ball nose cutters feature semi-finish to finish milling of the 3-D surfaces without a large machining allowance. The machined die and mold parts are not equal in their size and accuracy; they differ in processing chain and specification of rough and finish passes. Therefore the part manufacture makes different demands for the required tools in general and for the ball nose milling cutters specifically.

The chart in Fig. 48 shows in simplified form the type of a ball nose cutter as a function of the cutter accuracy and its nominal diameter. In some measures the chart can help in choosing the cutter: basing on the indexable-type principle the cutter can reach large diameters but its accuracy will be limited; at the same time a solid carbide endmill usually is the most suitable solution for high accuracy requirements; however the diameter of such a mill rarely exceeds 20 mm. The ball nose cutters of the single-insert type and the milling tools with the interchangeable solid carbide heads of the **MULTI-MASTER** family are typically found in-between the indexable and solid tools either in their accuracy or dimensional sizes. It is worthy to note that there are high-precision **MULTI-MASTER** ball nose heads with accuracy requirements more strict than for the solid mills, but it is more exception than the rule that says: the monolithic integral tool can be more precise than the assembled one.

Fig. 48



The starting cutting speed Vc and feed per tooth fz used for CNC programming can be found from the already known equations (1) and (6):

$$Vc = Vo \times Ks \times Kt \qquad (1)$$

$$fz = fz_0 \times KTH \times Ks \qquad (6)$$

Where: Vo – basic cutting speed
Kt – tool life factor
fz_0 – basic starting feed
KTH – chip thinning factor
Ks – stability factor

The tool life factor is shown in Table 8.

The chip thinning factor is defined by equation (16) or in Table 70.

The stability factor, as already noted, is taken to be 1 for normal conditions and to 0.7 if the estimated cutting stability is insufficient (milling thin walls, high overhang, poor toolholding, non-rigid workholding, etc.).

Defining the basic starting speed and feed depends on the type of a ball nose cutter and is connected with design features of the cutter family.

The subsections below emphasize the characteristic properties of the families of the ball nose cutter and contain the tables with the basic speeds and feeds necessary for cutting data calculation.

BALLPLUS and HELIBALL: families of single-insert ball nose milling cutters

A **BALLPLUS** milling cutter carries one ball nose insert with the V-shaped rear part that is mounted against two contact surfaces within the slot of the cutter body (Fig. 19). The cutter has two cutting edges which are the edges of the insert. The range of diameters for the **BALLPLUS** cutters is 12-25 mm.

The **HELIBALL** family, the immediate predecessor of the **BALLPLUS** and now almost completely replaced by it, is represented today only by the cutters of smaller diameters: 8-10 mm. Inserts for the cutters are shaped by sintering only without any grinding operation. A cutter has only one cutting edge in this case and is intended mostly for rough milling operations. The 10 mm insert, however, is indexable and there are two cutting edges on it; but the 8 mm insert, a typical throwaway, is provided with only one cutting edge.

The **BALLPLUS** inserts differ in angular values of their cutting edge, cutting geometry and accuracy (Table 71).

Table 71 **Ball Nose Inserts of the BALLPLUS Family**

Insert	Cutting Edge		Accuracy Grade	Rake Face	Flank	Application	
	Spheric	Cylindrical				Material type	Operation
HBR...	~220° (bulb-type)	no	normal	sintered	ground	wide range	rough to finish
HBF...			high	ground			finish
HCR...-QF	180° (hemisphere)	yes	normal	sintered			rough to finish
HCR...-QP						soft	

Basic starting feed fzo and basic cutting speed Vo are shown in Tables 72 and 73.

Table 72 **BALLPLUS and HELIBALL Ball Nose Milling Cutters: Basic Feed fzo, mm/tooth**

ISO class DIN/ISO 513	Material Group*	fzo, for D, mm					
		8	10	12	16	20	25
P	1-4	0.09	0.1	0.12	0.13	0.15	0.17
	5	0.08	0.09	0.11	0.12	0.14	0.16
	6, 7	0.08	0.09	0.1	0.12	0.13	0.15
	8, 9	0.07	0.08	0.1	0.11	0.12	0.13
	10	0.07	0.08	0.09	0.1	0.11	0.12
	11	0.06	0.07	0.09	0.1	0.1	0.11
M	12, 13	0.07	0.08	0.09	0.1	0.11	0.12
K	15-16	0.09	0.1	0.12	0.13	0.15	0.17
	17-18	0.08	0.09	0.11	0.12	0.14	0.16
H	38.1	0.05	0.06	0.07	0.07	0.08	0.1
	38.2	0.04	0.05	0.06	0.06	0.07	0.08
	39	0.03	0.03	0.04	0.04	0.05	0.06

* ISCAR material group in accordance with VDI 3323 standard

A rule of thumb

As mentioned earlier, in milling with ball nose cutters, the machining tolerance (stock) typically is not so large. For quick estimation of the depth of cut/width of cut relation, a rule of thumb may be used. The rule, named as "the rule of 12", gives quite acceptable results for milling soft and pre-hardened steel and martensitic stainless steel by ball nose cutters of one-insert type or with interchangeable solid heads. In accordance with the rule, if a depth of cut is the half of a cutter diameter (D/2), a width of cut should be no more than D/6; for the depth of cut D/3 the maximal width of cut should be D/4, etc.

It is not difficult to see that 2×6=3×4=12.

Table 73 **Ball Nose Cutters with BALLPLUS HCR.../HBR..., HELIBALL CR... and DROPMILL BCR... Inserts: Basic Cutting Speed Vo, m/min**

ISO Class DIN/ISO 513	Material Group*	Vc, for Grade		
		IC908	IC928	IC328
P	1	210	180	160
	2-4	200	170	140
	5-6	190	150	130
	7-9	180	140	125
	10	160	125	120
	11	140	120	110
M	12, 13	155	130	125
K	15-16	220	200	
	17-18	200	180	
H	38.1	100	80	
	38.2	70		
	39**	55		

* ISCAR material group in accordance with VDI 3323 standard
** In this case HSM recommended

Three carbide grades are available for the **BALLPLUS/HELIBALL** inserts of the standard line: IC908, IC928 and IC328. From these three grades, IC928 is conceptually the most general-duty and should be the first choice for milling soft and pre-hardened steel. However, due to the fact that the main part of the application of the one-insert ball nose cutters relates to semi-finish and finish milling, grade IC908, the hardest one, has become increasingly more popular, and can be considered to be the preferred option. In many cases it is the unique solution for profiling workpieces from hardened steels and nodular cast iron. The tougher grade IC328 is suitable for milling in unstable conditions and for cases with considerable impact loading. The fully shaped by sintering non-ground CR... inserts for 8 and 10 mm dia. **HELIBALL** tools are produced just from this grade.

Example

A part made from carbon steel AISI/SAE 1020 is machined by ball nose endmill cutter HCM D20-A-L150-C20, with insert HCR D200-QF IC908. The axial depth of cut is 5 mm, the width of cut (stepover) – 3.5 mm. Operational stability is sufficient.

Effective diameter De and chip thinning factor KTH can be calculated or found directly from Table 70: 17.3 mm and 1.1 respectively.
The workpiece material relates to the material group No.1.
Basic feed fzo=0.15 mm/tooth (Table 72).
Basic cutting speed Vo=210 m/min (Table 73).
Programmed feed per tooth fz=0.15×1.1= 0.16 (mm/tooth).
Cutting speed for estimated tool life 60 min.Vc=210×1×0.8=168 (m/min).
Spindle speed 1000×Vc/(π×De)=1000×168/(π ×17.3)=3090 (rpm).
Programmed feed speed V_F= 0.16×2×3090=988.8 (mm/min) ≈990 mm/min.

Ramping mathematics and one working rule

Chip thinning factor KTH, as a function of cutter diameter D and axial depth of cut ap, is defined by equations (16) and (4a). In ramping with ball nose cutters, ap often is not specified directly but it is defined implicitly via ramping angle α and machining allowance (stock to be removed) per part a. Of course, CAD/CAM systems allows very quick finding ap in such case. In addition, the axial depth of cut can be calculated from the following equation:

$ap=D/2+[(a-D/2)\times\cos\alpha+\sqrt{(D\times a-a^2)}\times\sin\alpha]$ (17)

If the phrase in the square brackets is positive, ap should be taken equal to the cutter radius or D/2.

In shop practice the result of calculation according to equation (17) is often increased by 10-20% (clearly, in any case the final value shall not exceed D/2). This working rule provides a sort of safety factor, making the programmed feed a little bit smaller. The safety factor takes into account non-uniform loading of a ball-shaped cutting edge during ramping.

Example

12 mm diameter ball nose endmill cutter HCM D12-D-L160-C16 with insert HBF D120-QF IC908 is applied to finishing an inclined bottom surface of a plastic mold part made from AISI P20 mold steel hardened to HRC 50-52. The operation is ramp up milling with a ramping angle of 5° and machining allowance of 0.2 mm. The cutter works with reasonably high overhang.

Find cutting data if the required tool life period is 20 minutes:

Effective diameter in accordance with Table 69, case b):
$De=(12-2\times0.2)\times\sin5°+2\times\sqrt{(12\times0.2-0.2^2)}\times\cos5°=4.1$ (mm).
Axial depth of cut (equation 17) $ap=12/2+[(0.2-12/2)\times\cos5°+\sqrt{(12\times0.2-0.2^2)}\times\sin5°]=0.36$ (mm).
Maximal cutting edge angle χmax = arccos (1-(2×0.2/12))= 14.8°.
Chip thinning factor KTH = 1/sin χmax=1/sin14.8°= 3.9.

The high tool overhang requires defining stability as poor and using stability factor Ks=0.7.

The machined material relates to material group No.38.2, hence:

- Basic feed fzo=0.06 mm/tooth (Table 72)
- Basic cutting speed Vo=70 m/min (Table 73)

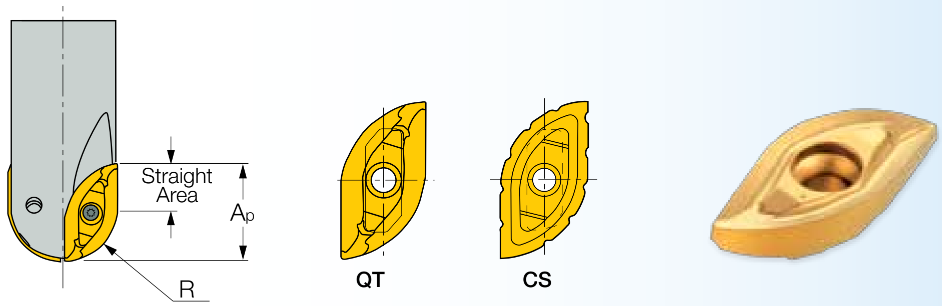
Programmed feed per tooth fz=0.06×3.9×0.7= 0.16 (mm/tooth)
Cutting speed for 20 min. tool life Vc=70×0.7×1=49 (m/min)
Spindle speed 1000×Vc/(π×De)=1000×49/(π ×4.1)=3804 (rpm)
Programmed feed speed V_F= 0.16×2×3804=1217 (mm/min)

Comparison of the cutting speed with respect to the nominal diameter will be as follows:
π×12×3804/1000=143 (m/min), while the real cutting speed is no better than 49 m/min.

DROPMILL: a Family of Indexable Ball Nose Milling Cutters

A **DROPMILL** cutter carries two indexable inserts of a teardrop shape, ensuring two fully effective teeth for the cutter (Fig. 49). The insert features two cutting edges and each of them combines a round corner with a straight side area tangent to the corner. The area has a relatively large length: 60-70% of a spherical radius generated by the corner, which enables machining a straight wall and a shoulder with a round corner. A protrusion from the insert bottom that fits into a matching slot in the insert pocket, allows the insert to withstand the high forces resulting from considerable loading.

Fig. 49



The main application of the family is for milling 3-D surfaces using various techniques, machining die and mold cavities by ramping by line or helix (helical interpolation), etc. The **DROPMILL** cutters are especially advantageous for semi-finishing the contoured surfaces; allowing for productive removal of significant machining stock. In addition, they are suitable for drilling operations with relatively small depth. The cutters are produced as integral tools and also as milling heads for modular tools with **FLEXFIT** or **MULTI-MASTER** connections.

There are two types of the **DROPMILL** inserts: the peripherally ground QT and fully sintered CS with chip splitting grooves. The latter is principally intended for heavy duty rough milling with large depths of cut, due to its chip slitting cutting effect that prevents the formation of long chips, reduces acting cutting forces and improves chip evacuation.

Table 74 **Ball Nose Inserts of DROPMILL Family**

Insert	Rake Face	Flank	Chip Splitter	Main Application	Tool Ø ,mm							
					12	16	20	25	30	32	40	50
BCR D…-QT	sintered	ground	no	semi-finishing								
BCR D…-CS		sintered	yes	roughing								

Similar to the families of the one-insert ball nose cutters, the **DROPMILL** inserts are produced from the same carbide grades: IC908, IC928 and IC328.

Table 73 contains data for basic cutting speeds. In milling with the **DROPMILL** cutters, a width of cut usually does not exceed 30% of the cutter diameter in rough operations and it is considerably less in semi-finish to finish passes. Therefore, in case of heavy-duty applications with a sizable width of cut or milling a full slot with a depth of cut more than a quarter of the diameter for the QT-type inserts and more than the third part of the diameter for the CS-type inserts, the table values should be reduced by 10-20%.

Table 75 **DROPMILL Ball Nose Milling Cutters: Basic Feed fzo, mm/tooth**

ISO Class DIN/ISO 513	Material Group*	fzo, for D, mm						
		12	16	20	25	30/32	40	50
P	1-4	0.11	0.13	0.15	0.17	0.18	0.21	0.25
	5	0.11	0.12	0.14	0.16	0.17	0.18	0.21
	6, 7	0.1	0.11	0.13	0.15	0.16	0.17	0.19
	8, 9	0.1	0.11	0.13	0.14	0.15	0.16	0.18
	10	0.09	0.1	0.12	0.13	0.14	0.15	0.17
	11	0.08	0.09	0.1	0.11	0.12	0.13	0.15
M	12, 13	0.09	0.1	0.12	0.13	0.14	0.15	0.17
K	15-16	0.11	0.13	0.15	0.17	0.18	0.21	0.25
	17-18	0.11	0.12	0.14	0.16	0.17	0.18	0.2
H	38.1	0.07	0.08	0.09	0.1	0.11	0.12	0.13
	38.2	0.05	0.06	0.07	0.08	0.09	0.09	0.1
	39	0.04	0.04	0.05	0.06	0.07	0.07	0.08

* ISCAR material group in accordance with VDI 3323 standard

Chip splitting drop

Milling slots or grooves (especially the deep ones by step passes), milling shoulders with large allowance, rough milling cavities and pockets with high stock removal rate – all these operations performed by the ***DROPMILL*** *cutters feature heavy tool loading and chip re-cutting.*
For such operations, using the CS-type inserts is the preferred way. The CS-type geometry should be considered as a first choice in any case when it comes to heavy-duty milling and problems with chip evacuation.

Basic feed per tooth is shown in Table 75. For milling a full slot with a depth of cut more than a quarter of the diameter for the QT-type inserts and more than the third part of the diameter for the CS-type inserts and for heavy-duty shoulder milling, the specified values should be reduced by 20-30%.

Example

A slot with a rounded bottom of 50 mm width and 43 mm depth is planned to be machined by ball nose endmill cutter BCM D50-A-W50-C with inserts BCR D500-CS IC908. The workpiece material is alloy steel AISI/SAE 6150, HB 250…260. The technological system has sufficient stiffness.
A process planner decided to mill the slot by three step passes: the first – with 25 mm depth of cut, the second – with 10 mm, and the third – with 8 mm depth of cut. Find cutting data:

The cutter radius is 25 mm. Hence, the effective diameter on the first pass will already be 50 mm; and chip thinning factor KTH = 1.

The machined material relates to material group No.8, therefore:

- Basic feed fzo=0.18 mm/tooth (Table 75)
- Basic cutting speed Vo=180 m/min (Table 73)

Taking into account the above remarks regarding milling a full slot we should reduce these values by 10-20% for the speed and by 20-30% for the feed. It means that on average the programmed feed per tooth will be 0.18×0.75=0.13 (mm/tooth); the cutting speed for 60 min. tool life - 180×0.85×0.8=122 (m/min).

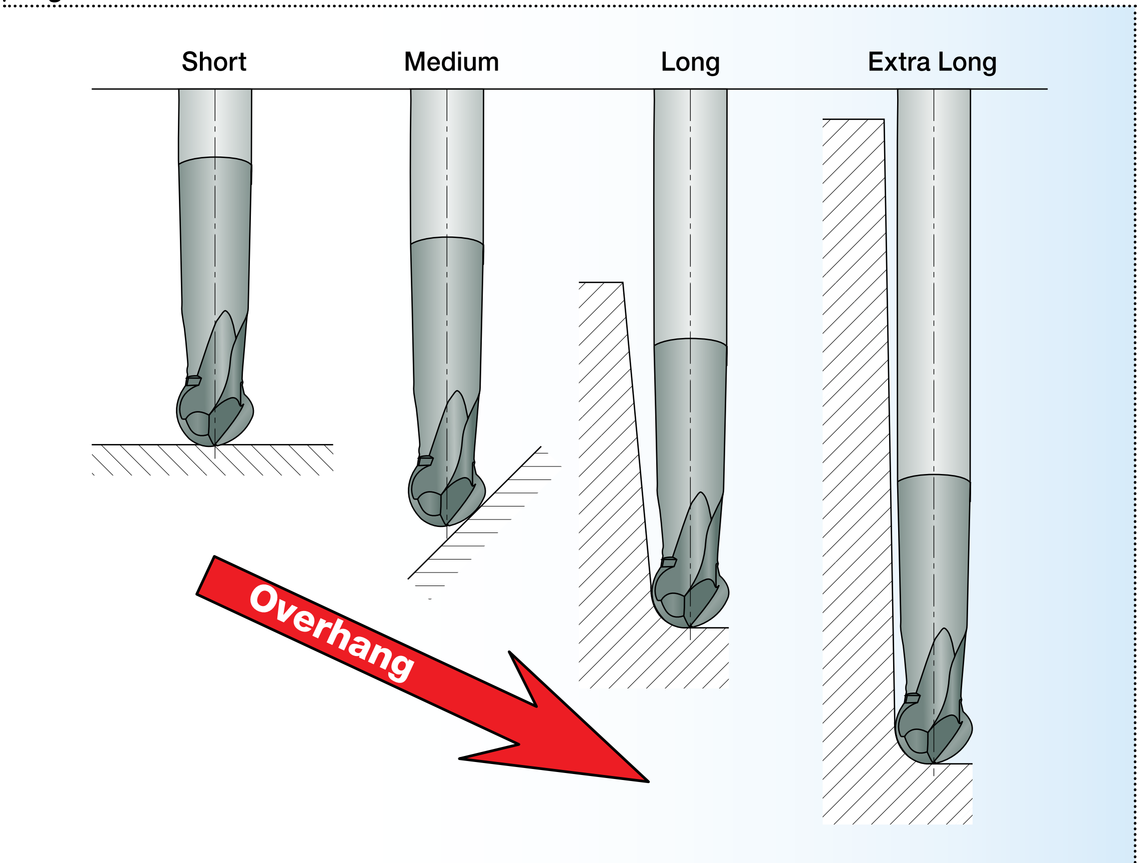
Spindle speed 1000×Vc/(π×De)=1000×122/(π ×50)=776 (rpm) ≈ 770 rpm
Programmed feed speed V_F= 0.13×2×776=202 (mm/min) ≈ 200 mm/min

Remark
We took the average factor for reducing the speed and feed. A more correct approach demands less decrease for the first pass and greater reduction for the third, the last one because the chip evacuation becomes problematic. Let say, 135 m/min with 0.14 mm/tooth for the first pass and 115 m/min with 0.12 mm/tooth for the third pass.

Solid Carbide Ball Nose Endmill Cutters

SOLIDMILL, the ISCAR line of solid carbide endmills, contains ball nose cutters varying in cutting geometry and number of flutes. The cutters are produced from various carbide grades (Table 10) in short, medium, long and extra long length versions (Fig. 50), with straight or tapered neck and intended for all types of the die and mold materials. Generally, the range of cutting diameters for the solid carbide ball nose endmill tools is 1 …25 mm (0.4…2 mm for miniature cutters); and the helix angle is 30°. The diameter has accuracy tolerance e8 (h10 for the economical **SOLIDECO** family, Table 76).

Fig. 50



Obviously, an integral solid cutter provides a more accurate machining solution relative to an indexable mill. Therefore, the main application of the solid carbide ball nose cutters relates to finish and semi-finish milling of the 3-D surfaces; and they primarily operate under small machining allowance or stock (Fig. 45), but the cutters are used successfully in roughing applications too. In machining with low allowances, even a rough stock is small enough; and the borders between different types of machining are not so clear. Table 77 can help in estimation of the type of machining for main die and mold materials and corresponding machining allowance.

Two or four?

Classically the ISCAR solid carbide ball nose milling cutters have 2 or 4 flutes, that is - 2 or 4 cutting teeth. How to choose the number of flutes for a tool?
The all-purpose 4 flute ball nose cutters give a universal robust and productive solution for various applications, especially for semi-finish and finish milling. Oppositely, the 2 flute cutters with greater chip gullet are more suitable for rough milling, ensuring better chip evacuation. In addition, using the 2 flute cutters is a workable method for fine finishing due to less accumulated error, which depends on the number of teeth.
Concerning the 4 flute cutters, traditionally only two teeth produce the central cutting area near the cutter tip and not four. Then in finish milling with shallow depth of cut, calculation of the feed per tooth should take into consideration only two effective teeth; and the advantages of a multi-flute cutter are diminished. The 2 flute tool is a more preferable means for such a case. Also, it prevents cutting action by a "conversion zone" between 2 and 4 teeth, specific for the 4 flute tools that is marked by intensive wear of the non-central teeth and accuracy errors.

Table 76 **Solid Carbide Ball Nose Milling Cutters**

Cutter	Cutting Edge		Z*	Diam. Range, mm	ECO Line	Carbide Grades		Main Milling Application	Workpiece Hardness
	Spherical	Cylindr.				Standard	ECO		
EBRF	180°	yes	3; 4	6…20	no	IC903	–	roughing	up to HRC 55
EB	180°	yes	2; 3; 4	0.4…25	yes	IC903	–	hard materials	up to HRC 65**
						IC900; IC300; IC08	IC900; IC08	general-duty	up to HRC 45
ESB	~220°	no	2; 4	3…16	no	IC903	–	hard materials	up to HRC 65

* Number of flutes
** A subgroup of 2 flute EB …A…cutters is intended for materials with HRC 55…70

Table 77 **Solid Carbide Ball Nose Cutters and MULTI-MASTER MM EB… Ball Nose Heads: Type of Machining and Approximate Machining Allowance**

ISO Class DIN/ISO 513	ISCAR Mat. Group*	Machined material		Type of machining		
				Roughing	Semi-finishing	Finishing
P	1, 2, 4, 6, 10	soft steel	HB<250	0.12×D**	0.07×D	0.02×D
	3, 7		HB 250…300	0.1×D	0.05×D	0.015×D
	5, 8, 9, 11	pre-hardened steel	HRC 30-37	0.1×D	0.05×D	0.01×D
	9-11		HRC 38-44	0.08×D	0.04×D	0.01×D
M	12, 13	martensitic s.s.	HB<250	0.1×D	0.05×D	0.01×D
K	15-18	cast iron	HB< 300	0.15×D	0.08×D	0.03×D
H	38.1	hardened steel	HRC 45-49	0.08×D	0.04×D	0.01×D
	38.2		HRC 50-55	0.06×D	0.04×D	0.01×D
	39		HRC 56-63	0.05×D	0.03×D	0.01×D

* ISCAR material group in accordance with VDI 3323 standard
** D - the diameter of a ball nose cutter (head)

Basic cutting speeds and feeds can be defined from Tables 78 and 79. As already stated, the only carbide grade should be chosen from Table 10.

Table 78 **Solid Carbide Ball Nose Cutters and MULTI-MASTER Ball Nose Heads: Basic Cutting Speed Vo***

ISO Class DIN/ISO 513	ISCAR Mat. Group**	Vo, m/min, for Type of Machining		
		Roughing	Semi-finishing	Finishing
P	1	180	220	280
	2-4	150	170	200
	5	125	140	170
	6	130	150	190
	7-9	120	135	170
	10	115	130	165
	11	100	110	120
M	12, 13	110	110	150
K	15-16	160	180	220
	17-18	150	170	200
H	38.1	70	80	100
	38.2		40	50
	39		30	40

* Carbide grade selector for solid cutters – refer to Table 10
** ISCAR material group in accordance with VDI 3323 standard

Example
It is suggested to mill a master hob, which is made from AISI A6 tool steel hardened to HRC 55, on a vertical machine center, using a 12 mm diameter solid carbide ball nose cutter EB-A2 12-12/24C12H110 903. The hob walls are inclined 3°.
Machining stock to be removed during this milling operation is 0.1…0.15 mm on side.
The machine is in a good condition; the hob is properly clamped into a fixture on the machine table.
Find starting cutting data.

According to Table 69, case b) effective diameter De:
$De=(12-2\times0.15)\times\sin3°+2\times\sqrt{(12\times0.15-0.15^2)}\times\cos3°=3.3$ (mm)
Axial depth of cut ap can be readily calculated (Fig. 47) as:
$ap=D/2-0.5\times\sqrt{(D^2-De^2)}=12/2-0.5\times\times\sqrt{(12^2-3.3^2)}=0.23$ (mm)
From Table 70 chip thinning factor KTH≈3.5.

Consequently, for material group No. 38.2, to which relates the machined steel, the basic cutting speed is 50 m/min (Table 78); the basic feed per tooth is 0.011 mm/tooth (0.013×0.85 – refer to Table 79 and remarks to it).

Programmed feed per tooth fz=0.011×3.5×1= 0.038 (mm/tooth).
Cutting speed for 20 min. tool life Vc=Vo=50 m/min.
Spindle speed 1000×Vo/(π×De)=1000×50/(π ×3.3)=4822 (rpm).
Programmed feed speed V_F= 0.038×2×4822=366 (mm/min).

It is interesting that the cutting speed calculated relative to the nominal diameter of the cutter (12 mm) will be: VcD =π×12×4822/1000=182 (m/min)

Reminder
Always check how a report, technical paper or guide specifiies a cutting speed, with respect to effective or nominal diameter (Fig. 51)!

Fig. 51

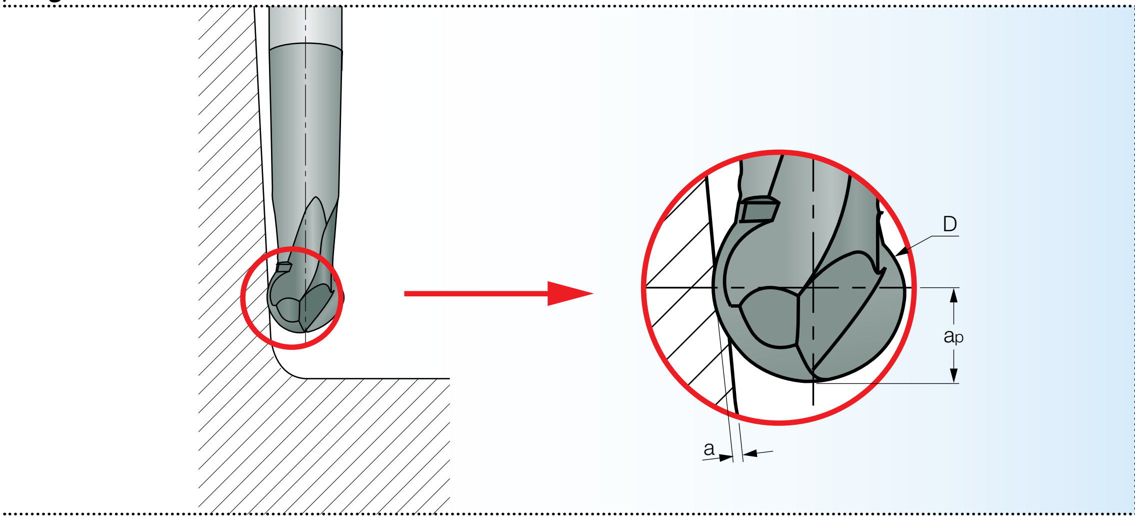


Table 79 **Solid Carbide Ball Nose Cutters and MULTI-MASTER MM EB… Ball Nose Heads: Basic Feed fzo***

ISO Class DIN/ISO 513	ISCAR Mat. Group**	fzo, mm/tooth, for D***											
		1	2	3	4	5	6	8	10	12	16	20	25
P	1-4	0.008	0.02	0.03	0.04	0.05	0.055	0.062	0.07	0.08	0.09	0.11	0.13
	5	0.007	0.018	0.025	0.032	0.038	0.043	0.05	0.06	0.068	0.075	0.085	0.11
	6, 7	0.007	0.018	0.02	0.028	0.035	0.04	0.048	0.057	0.062	0.072	0.082	0.09
	8, 9	0.007	0.015	0.02	0.025	0.032	0.035	0.038	0.045	0.05	0.06	0.07	0.08
	10	0.007	0.013	0.015	0.022	0.027	0.03	0.032	0.038	0.042	0.05	0.06	0.07
	11	0.006	0.009	0.01	0.015	0.02	0.025	0.028	0.035	0.038	0.045	0.055	0.06
M	12, 13	0.007	0.013	0.015	0.022	0.027	0.03	0.032	0.038	0.042	0.05	0.06	0.07
K	15-16	0.008	0.021	0.031	0.042	0.052	0.058	0.065	0.073	0.085	0.095	0.117	0.137
	17-18	0.007	0.021	0.031	0.042	0.052	0.058	0.065	0.073	0.085	0.095	0.117	0.137
H	38.1	0.006	0.007	0.008	0.012	0.017	0.021	0.025	0.027	0.03	0.035	0.04	0.05
	38.2	0.005	0.005	0.006	0.007	0.007	0.008	0.009	0.01	0.013	0.015	0.02	0.025
	39	0.004	0.004	0.005	0.005	0.006	0.007	0.007	0.008	0.009	0.009	0.01	0.013

* The table reflects rough to semi-finish milling operations
For semi-finish to finish applications, the table values need to be reduced by 10-20%
Carbide grade selector for solid cutters – refer to Table 10

** ISCAR material group in accordance with VDI 3323 standard

*** D - the diameter of a cutter (head), mm
For cutters with the diameter less than 1 mm, refer to corresponding ISCAR user guides and catalogs

MULTI-MASTER: Interchangeable Ball Nose Milling Heads

The **MULTI-MASTER** family offers a rich choice of ball nose milling heads with various dimensions, shapes and accuracy. The heads combined with shanks, extensions and reducers allow for assembly of numerous ball nose endmill cutters that meet the requirements of the die and mold maker.

Similar to the **MULTI-MASTER** head shapes (square, toroidal, etc.), there are two main types of the ball nose heads: multi-flute MM EB… heads and two flute "economy" MM H… heads (Table 80).

The cutting geometry of MM EB… heads is not different from the solid carbide ball nose cutters; however the heads usually have smaller cutting lengths. The approach to finding cutting data for MM EB… heads is the same as the method for the solid cutters.

Table 80 **Ball Nose Heads of MULTI-MASTER Family**

Head	Type	Cutting Edge		Rake Face	Flank	Z*	Ø tol.	Carbide Grade	Head Ø, mm						Main Milling Application
		Spherical	Cylindr.						6	8	10	12	16	20	
MM HCR	economy (pressed to size)	180°	yes	sintered	ground	2	h9	IC908							rough to finish
MM HRF				ground			h7	IC903							finish + milling hard steel
MM HBR		~240°	no				h7	IC908							
MM EB	ground	180°	yes		ground	2; 4	e8	IC908							rough to finish

* Number of flutes

MM H… ball nose heads are called "economy", but not due to lesser accuracy. On the contrary, high-precision MM HBR… and MM HRF… heads have very closed diameter tolerances (h7). The heads, as other pressed and sintered "to shape and size" heads of this type, require minimum grinding operation during their manufacturing. Such technology allows for unique features: the combination of negative rake angles near the head tip with positive rake angles along the most part of the spheric cutting edge in MM HCR… heads, increased area of the spheric edge in bulb-type MM HBR… heads and others. A tooth of the head is extremely strong and successfully stands up against considerable loading; which is important for rough profiling and milling hardened steel. However, the helix angle of the heads is much less when compared with the solid carbide ball nose cutters and MM EB… heads.

The **MULTI-MASTER** ball nose heads are made from carbide grade IC908, except for the heads MM HRF… intended for precise finishing hardened steel, for which carbide grade IC903 is used.

Concerning cutting data, Table 81 shows average machining allowance for different types of milling with the use of **MULTI-MASTER** ball nose tools carrying MM H… heads. The basic cutting speed can be found from Table 78 and the basic feed per tooth – from Table 82.

Example
Within a planned process for manufacturing a die casting die, a ball nose cutter, which comprises shank MM S-B-L125-C16-T06 assembled with ball nose head MM HCR100-2T06 IC908, is the subject of restmilling the corners of a part of the die. The part material is tool steel AISI H11 with hardness HRC 40…42. The restmilling cycle consists of several passes with machining allowance 0.2 mm max. Operational stability is good. Cutting data must be found.

Assume that all cutting edges perform cutting and the effective diameter De is equal to the nominal diameter (10 mm) – the worst case. Then chip thinning factor KTH = 1.

The machined material represents material group No.10.

Table 81 **MULTI-MASTER MM H… Ball Nose Heads: Type of Machining and Approximate Machining Allowance**

ISO Class DIN/ISO 513	ISCAR Mat. Group*	Machined Material		Type of Machining		
				Roughing	Semi-finishing	Finishing
P	1, 2, 4, 6, 10	Soft steel	HB<250	0.14×D**	0.08×D	0.02×D
	3, 7		HB 250…300	0.12×D	0.06×D	0.015×D
	5, 8, 9, 11	Pre-hardened steel	HRC 30-37	0.11×D	0.05×D	0.01×D
	9-11		HRC 38-44	0.09×D	0.04×D	0.01×D
M	12, 13	Martensitic s.s.	HB<250	0.11×D	0.05×D	0.01×D
K	15-18	Cast iron	HB< 300	0.17×D	0.08×D	0.03×D
H	38.1	Hardened steel***	HRC 45-49	0.09×D	0.04×D	0.01×D
	38.2		HRC 50-55	0.06×D	0.04×D	0.01×D
	39		HRC 56-63	0.05×D	0.03×D	0.01×D

* ISCAR material group in accordance with VDI 3323 standard
** D - the diameter of a head
*** Recommended carbide grade - IC903

Restmilling

As a rule, productive milling passes, especially rough, propose applying more durable and rigid tools for high metal removal rate. In many cases the form and dimensions of the tool do not allow for a cut in some areas; for example, the corners of a die cavity (Fig. 52). The remainder of the material in the areas is removed by restmilling – a method under a technological process where a tool of smaller diameter cuts the areas with residual stock. In speaking about restmilling, this is a finishing operation that is a part of the whole finish milling cycle.
Making use of the solid carbide ball nose cutters and the ***MULTI-MASTER*** *ball nose heads is the excellent choice for restmilling tools.*

Fig. 52

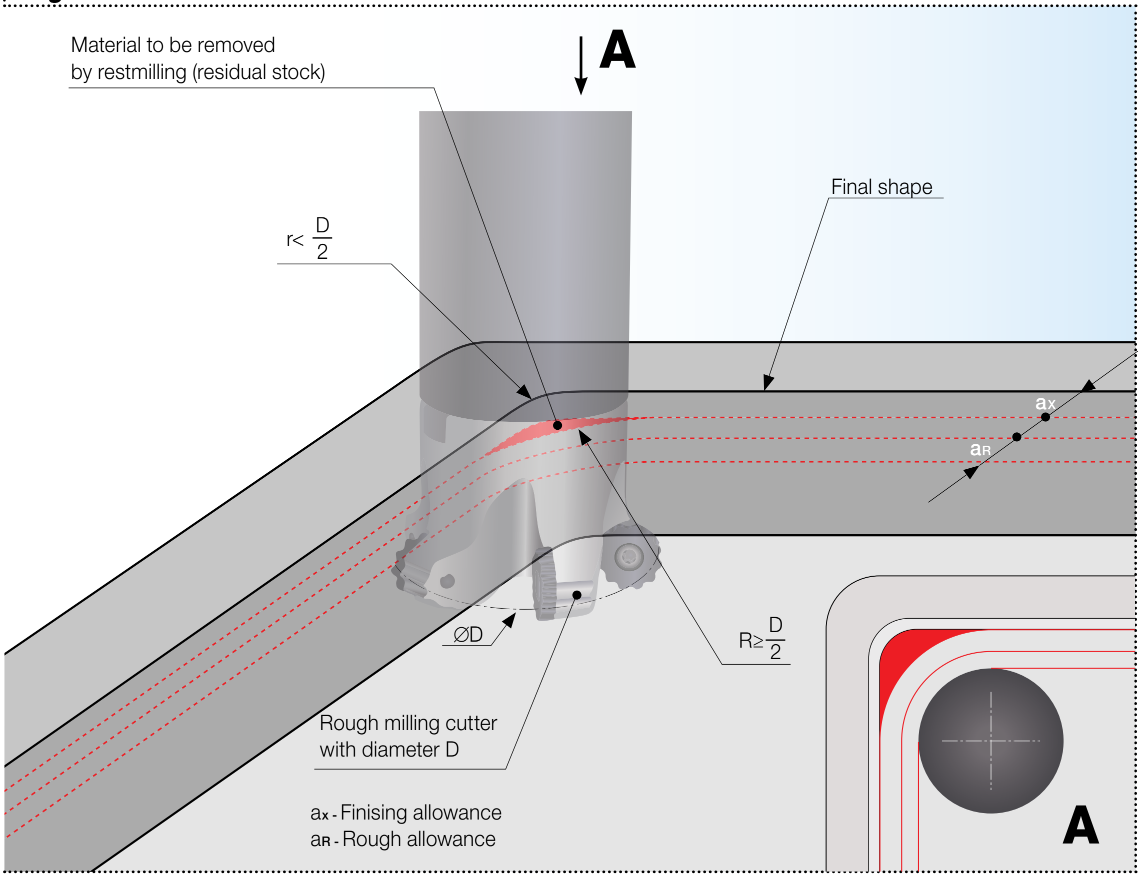


Table 81 **MULTI-MASTER MM H… Ball Nose Heads: Basic Feed fzo***

ISO class DIN/ISO 513	ISCAR Mat. Group**	fzo, mm/tooth, forD***				
		8	10	12	16	20
P	1-4	0.08	0.09	0.11	0.12	0.13
	5	0.07	0.08	0.1	0.11	0.12
	6, 7	0.07	0.08	0.09	0.1	0.11
	8, 9	0.06	0.07	0.08	0.09	0.1
	10	0.06	0.07	0.08	0.09	0.1
	11	0.05	0.06	0.07	0.08	0.09
M	12, 13	0.06	0.07	0.08	0.09	0.1
K	15-16	0.08	0.09	0.1	0.12	0.13
	17-18	0.08	0.09	0.1	0.11	0.12
H	38.1	0.04	0.05	0.06	0.07	0.08
	38.2	0.03	0.037	0.045	0.055	0.065
	39	0.02	0.025	0.03	0.035	0.04

* The table reflects rough to semi-finish milling operations
For semi-finish to finish applications the table values need to be reduced by 10-20%
For roughing by a cutter with MM HBR…head, the table values need to be reduced by 10%
For machining hardened steel - carbide grade IC903 is recommended
** ISCAR material group in accordance with VDI 3323 standard
*** D - the diameter of a head

Comparing the machining allowance with the values of Table 81, the operation fits into finish milling. So the basic cutting speed for 20 min. tool life is 165 m/min (Table 78); the basic feed per tooth is 0.06 mm/tooth (Table 82: table value 0.07 mm/tooth with reducing by 10% in accordance with the remark).
Spindle speed 1000×Vo/(π×De)=1000×165/(π ×10)=5252 (rpm)
Programmed feed speed V_F = 0.06×2×5252=630 (mm/min)

Die and Mold

3.4. Special Methods: High Speed Machining (HSM) and Milling of Hardened Steel

The term "High Speed Machining" and the abbreviation "HSM" are well-known in industry, and in particular, in die and mold making. High speed machining, what is it? Why is HSM so relevant for up-to-date die and mold manufacturing?

Even up to now a generally accepted definition of high speed machining does not exist. Obviously, HSM can mean:

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

HSM is often emphasized that it is "…a high-efficiency method of modern machining with high spindle and feed speeds…" that allows for various advantages.
It is clear that cutting, spindle and feed speeds are interrelated. More spindle speed = more feed speed, etc. How to define the cutting speed?

In the example of machining steel with hardness HRC 55 to the section "Solid carbide ball nose cutters" the real cutting speed related to the effective diameter was 50 m/min, and the speed with respect to the nominal diameter – 182 m/min. More illustrative is a case of milling with 0.1 mm depth of cut by ball nose cutters of different diameters (Table 83). The table shows distinctly that in profile milling with shallow depths of cut, even small cutting speeds can require considerable spindle speed, which in turn causes a cutter to run fast.
Assume that 4 mm diameter ball nose cutter in Table 83 has 2 teeth and it cuts with feed 0.04 mm/tooth. The feed speed will be 815 mm/min, 1222 mm/min and 1630 mm/min for cutting speed 40 m/min, 60 m/min and 80 m/min correspondingly!

Shallow, light cuts combined with high spindle speeds are classical attributes of high speed machining. HSM, as a metalcutting method, demands very specific cutting techniques and dedicated machine tools, toolholding devices and of course, cutting tools, that will ensure acceptable tool life. Being applied to manufacturing process HSM can dramatically cut cycle time.
It is worthy of note that HSM not always a process with high spindle speed; and many HSM operations do not need high rotational velocity.

History: HSM

In the twenties-thirties Dr.-Ing. Carl Salomon, a German researcher, did a series of experiments for measuring cutting temperatures against the corresponding cutting speeds during machining of some engineering materials. The results, which have been represented graphically, allowed the researcher to assume that the cutting temperature rises with increasing the cutting speed, until the speed reaches some specific critical value, and then comes down, in spite of further the speed growth (Fig. 53). Hence, Salomon concluded, there is an area of cutting speeds much more than usual for which the cutting temperature is similar to those that are observed in conventional cutting. By hypothesis the area related to speeds is 5-10 times as much as common values.

Fig. 53

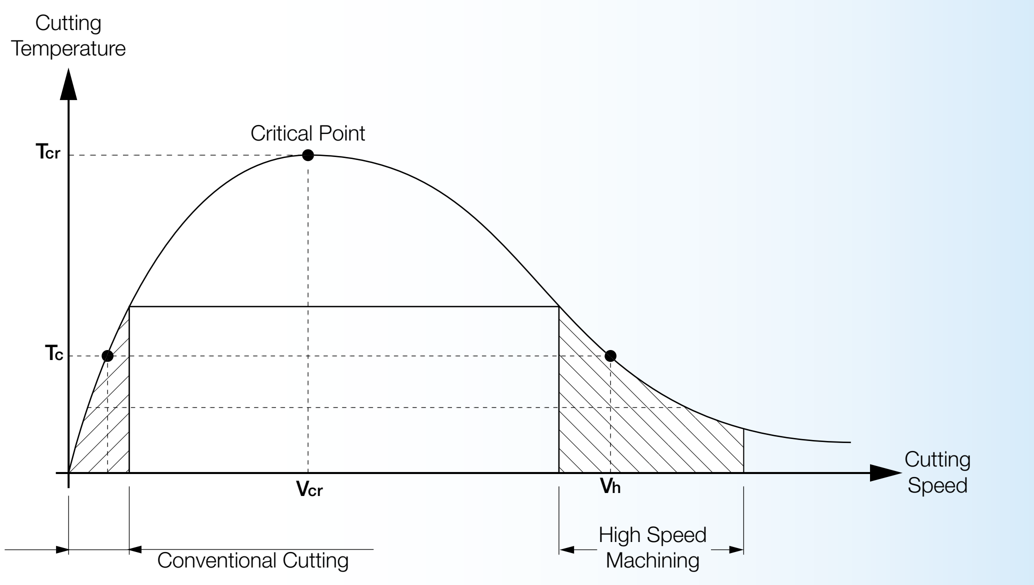


Table 83 **Comparative Example: Milling by Ball Nose Cutters of Different Diameters with 0.1 mm Depth of Cut**

Vc, m/min	D, mm											
	SØ 20		SØ 10		SØ 8		SØ 6		SØ 4		SØ 2	
	De= 2.8		De= 2		De= 1.77		De= 1.5		De= 1.25		De= 0.87	
	rpm	Vc_D	rpm	Vc_D	rpm	Vc_D	rpm	Vc_D	rpm	Vc_D	rpm	Vc_D
40	4547	286	6366	200	7193	181	8488	160	10186	128	14635	92
60	6820	429	9549	300	10789	271	12732	240	15279	192	21952	138
80	9094	572	12732	400	14386	362	16976	320	20372	256	29270	184

D - the diameter of a ball nose cutter
De, mm, - the effective diameter for 0.1 mm depth of cut
Vc - the cutting speed with respect to effective diameter De
rpm - spindle speed
Vc_D, m/min, - the cutting speed with respect to diameter of a cutter D

Today, HSM has penetrated deeply enough mainly into the following processes: machining of aluminum in aerospace or automotive industries and steel in die and mold making. Exactly in the die and mold branch, HSM advantages led to drastic changes in the entire manufacturing approach.

Traditionally, die and mold making expected a considerable share of cutting operations on soft steel, electro discharge machining (EDM) and finishing on hardened steel; and then manual smooth finish and polishing. EDM and manual finish and polish are very time-consuming processes. Repeatability of manual work is a very low likelihood, which presents additional problems for production. Intense competition on the die and mold markets dictates that manufacturers cut cycle time as much as possible and eliminate manual finish and polish. Success in CNC control, high-speed spindle technology, drives, bearings and other components of machine tool, balanced high-precision toolholders and of course, new cutting tool materials and geometries opened to HSM a road to the die and mold maker. The HSM method was very successful. It changed die and mold making processes, allowing for starting machining of already hardened workpieces, and left manual finish and polishing out; cutting the cycle time by 40-50%. Therefore HSM high speed machining in die and mold practice, connected first of all with high speed milling of hardened steel and especially by tools with small nominal diameters.

Hard steel

Hardened steel is at the same time hard-to-machine steel. Not so long ago a workpiece of hardness HRC 45 was considered to be the limit for metal cutting; and for more hard materials grinding operations should be applied. Progress in cutting tool technology seriously shook the understanding of the hardness limits. Today "hard steels" often refers to steel hardened to HRC 60 and more. In the context of such steels HSM is an effective instrument that makes their cutting possible.

In machining hardened steels - which are difficult-to-cut materials - intensive heat generation and vibration take place. It is a source of poor tool life, reduction of accuracy, loss of stability, etc. that makes machining operations unpredictable. Such vagueness is simply unacceptable for die and mold manufacturing, especially for large-sized die and molds that sometimes need to be cut during several days' time. The HSM technique gave the die and mold maker a good tool for overcoming this problem. Of course, HSM is not only cutting tools but also dedicated machine tools, control systems and toolholding devices – the combination of all the means together results in real advantages.

Conceptually, HSM of hardened steel is an integrated technological method for which every one of the involved components: the machine tool, the CNC hardware and software, the toolholder and of course the cutting tool, is a mandatory participant and a critical part of the whole process. It differs essentially from milling hardened steel under conventional conditions and from HSM of soft steel. Therefore, defining a more suitable tool and starting cutting data require more deep examination; and we recommend contacting ISCAR specialists in this field. For initial estimation and a general knowledge of the matter, Table 84 below contains typical average data referring to the square end solid carbide and **MULTI-MASTER** cutters with diameter 4…25 mm.

A rule of thumb

In HSM of hardened steel, starting feed per tooth feed per tooth is often estimated as 0.5-1% of a cutter diameter. The lower feed values correspond to more hard steel.

Table 84 **Average Starting Cutting Data for HSM of Steel Parts**

Steel Conditional Group		**Pre-hardened**	**Hardened**			
Material Group*		**11**	**38.1**	**38.2**	**39.1**	**39.2**
Hardness, HRC		**38-44**	**45-49**	**50-55**	**56-60**	**>60**
Rough Milling	ae	(0.40-0.70)D	(0.40-0.60)D	(0.40-0.60)D	(0.35-0.55)D	(0.30-0.50)D
	ap	(0.07-0.15)D	(0.06-0.12)D	(0.05-0.08)D	(0.04-0.07)D	(0.03-0.07)D
	Vc, m/min	120-190	100-170	90-150	70-100	60-80
	fz, mm/tooth	0.03-0.24	0.03-0.22	0.03-0.2	0.03-0.17	0.02-0.14
Semi-finish Milling	ae	(0.40-0.70)D	(0.40-0.60)D	(0.35-0.55)D	(0.35-0.45)D	(0.30-0.45)D
	ap	(0.05-0.12)D	(0.04-0.09)D	(0.03-0.07)D	(0.03-0.05)D	(0.02-0.05)D
	Vc, m/min	140-200	130-180	120-170	90-150	70-130
	fz, mm/tooth	0.03-0.22	0.03-0.2	0.03-0.17	0.03-0.14	0.02-0.12
Finish Milling	ae	(0.40-0.70)D	(0.40-0.55)D	(0.35-0.50)D	(0.35-0.45)D	(0.30-0.40)D
	ap	(0.04-0.1)D	(0.03-0.08)D	(0.02-0.06)D	0.1-0.4 mm	0.1-0.3 mm
	Vc, m/min	150-230	180-220	170-190	130-200	110-160
	fz, mm/tooth	0.03-0.2	0.03-0.18	0.02-0.15	0.02-0.13	0.015-0.11

* ISCAR material group in accordance with VDI 3323 standard
D - cutter diameter
ae - width of cut (stepover)
ap - depth of cut (stepdown)
Vc - cutting speed with respect to effective diameter De

Reminder

Some technical papers and advertising leaflets related to HSM and in particular to HSM of hardened steel, use cutting speed data which is not actual. Please check thorogly which diameters, effective or nominal these values refer to.

Example

In a die and mold shop a technologist plans to change the process of manufacturing a cavity plate by introducing HSM in the final machining steps. The technologist checks a proposed process by test milling. In accordance with the process, the plate, which is made from AISI O1 tool steel, will be machined in hardened condition (HRC 58…60) by solid carbide endmill EC-A4 060-13C06-50 IC903 on a recently acquired new high speed machining center. Find the starting cutting data for test milling passes.

We have a good basis to assume that machining conditions are stable. Taking into account that Table 84 relates mostly to typical data for cutters within diameter range 4…25 mm and the mill diameter is 6 mm, we can extract from the table the following starting data referring to finish milling for ISCAR material group No. 39.1:

Width of cut = 0.4×6=2.4 mm

Depth of cut = 0.15 mm

Cutting speed = 130 m/min

Feed = 0.03 mm/tooth

Spindle speed = 1000×130/(π×6)=6896 (rpm)≈7000 rpm

Feed speed = 0.03×4×7000=840 (mm/min)

Remark

The spindle speed lays within boundaries of "conventional" values. However, do not forget that the above case refers to machining tool steel with high hardness, for which "normal" cutting speed (and correspondingly, spindle speed) is twice as little!

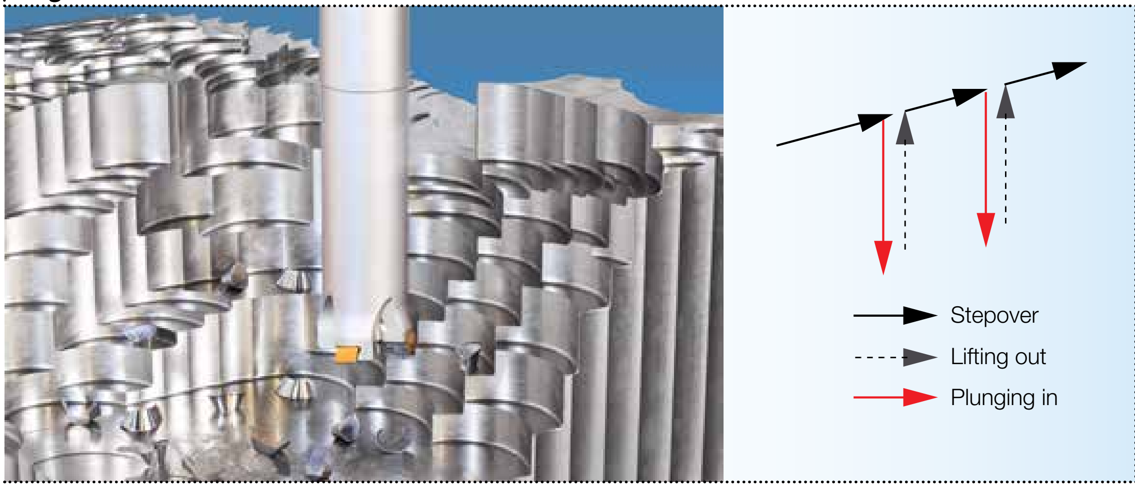
Keep in mind

Various research shows that for HSM with the use of a ball nose cutter, the best surface finish can be reached when the stepover will be equal to the feed per revolution..

3.5. Special Methods: Sculpturing by Plunge Milling

Plunge milling or simply "plunging", is a productive rough milling method during which a tool moves directly downward into a workpiece with feed directed along the tool axis (Fig. 54). It looks like drilling: the tool plunges into the workpiece. But despite the fact that some plunge milling tools (usually called plungers) are able to produce holes, their main function is to machine straight walls or slots.

Fig. 54



In plunging, the tool classically plunges in, lifts out, moves stepwise linearly and then continues this cycle again. It is clear that this machining technique produces a serrated surface with cusps that can be reduced by decreasing a linear stepover; but nevertheless the surface usually requires additional milling for better finish.

Plunging had spurted into popularity among die and mold makers due to one important feature: the main advantage of plunging is the ability to machine at high overhang with minimal bending forces. When high overhang is applied on a conventional milling cutter, which removes material layer-by-layer in passes, the cutter and its adapter quite often can not withstand the great bending moments because of the considerable radial component of cutting forces. Furthermore, the spindle of a machine tool takes up the most part of the radial forces. The result is vibration, noise and the cutter deviation, which causes inaccurate machining and damage of the tool, the adapter and the machine tool. Vibration and noise automatically lead to reducing cutting data and decreasing productivity.

In plunging, the main component of the cutting force acts axially – exactly in the direction of the highest rigidity of a machine tool – allowing productive milling and excellent straightness of machined surfaces.

In addition to plunging ability, the main intended purpose of the plunge milling cutters (also called plunge-in cutters or plungers), is often to perform face milling with limited depth of cut for obtaining a flat surface at bottoms and HFM at high overhang, boring operation, etc.

ISCAR offers a great variety of the plunge milling cutters. Table 85 shows general characteristics of the more popular representatives.

Plunge-in cutting options

Plunge-in capacity is not a feature incidental to plungers. Some other milling cutters (for example, ***MULTI-MASTER*** *tools with toroidal heads MM HT…) can also perform this milling technique. However, their use as plunging tools has some limitations and should be considered as only optional.*

Table 85 **Main ISCAR Plunge Milling Cutters**

Family	Designation	Dia. Range mm	Insert Clamping	Configuration			Main Features	Options		
				Endmill	ShellMill	FLEXFIT*		Face Milling	HFM	Ramping
TANGPLUNGE	HTP …LN10	25…52	Tangential	■	■	■	Side Plunger	■		
	HTP …LN16	50…100	Tangential		■			■		
HELITANG	FTP …LN10	50…63	Tangential		■				■	■
PLUNGER	PH …-13	40…63	Tangential		■	■	Center Cutting Side Plunger	■		
	PLX …-12	32…80	Radial		■					

* Heads with **FLEXFIT** or adaptation

Plunge milling is an effective and economical method for machining deep cavities, walls, slots and shapes. At the same time, correct machining strategy and CNC programming for plunging have unique features. Therefore we strongly advise studying corresponding guides or technical papers and/or to contact an ISCAR milling specialist in order to plan your process properly.

However, for better understanding of the subject, the explanations and tables below specify initial cutting data for the most popular side plungers of the HTP …LN-type from the TANGPLUNGE family.

Plunging sculptor

Axial movements of a plunging cutter bring to mind a working stone chisel in stone sculpturing (especially on its early, rough stages), which also plunges into material in a series and removes it. That plunge milling is sometimes called "sculpturing" and a 3-D shape obtained by plunging is called the "sculptured surface" (Fig. 55, but be careful – in general every 3-D shape also can be called "sculptured").

Fig. 55

Starting cutting speed Vc and feed per tooth fz can be found from the following equations (17) and (18):

$$Vc = Vo \times KH \times Kt \qquad (17)$$

$$fz = fzo \times KH \qquad (18)$$

Where: ***Vo*** – basic cutting speed for 20 min. tool life (Table 88)
KH – overhang coefficient (Table 86)
Kt – tool life factor (Table 8)
fzo – basic starting feed (Table 89)

Table 86 **Overhang Coefficient KH for the TANGPLUNGE Cutters as the Function of Ratio of Overhang H to Cutter Diameter D**

H/D	to 4	over 4 to 6	over 6 to 8	over 8 to 10
KH	1	0.9	0.8	0.7

Table 87 **TANGPLUNGE HTP...LN Plungers:**
Maximal Stepover L1max and Width of Plunge ae for Cutter Diameter D* (Fig. 56)

ae, mm	D, mm															
	HTP... 06			HTP...10								HTP...16				
	16	20	25	25	32	35	40	42	50	52	50	52	63	66	80	100
	L1max, mm															
1	7.7	8.7	9.8	9.8	11.1	11.7	12.5	12.8	14	14.3	14	14.3	15.7	16.1	17.8	19.9
2	10.5	12	13.5	13.5	15.4	16.2	17.4	17.8	19.5	20	19.5	20	22	22.6	24.9	28
3	12.4	14.2	16.2	16.2	18.7	19.6	21	21.6	23.8	24.3	23.8	24.3	26.8	27.5	30.4	34.1
4	13.8	16	18.3	18.3	21.2	22.3	24	24.6	27.1	27.7	27.1	27.7	30.7	31.4	34.8	39.1
4.5	14.3	16.7	19.2	19.2	22.2	23.4	25.2	25.9	28.6	29.2	28.6	29.2	32.4	33.2	36.8	41.4
5		17.3	20	20	23.3	24.5	26.5	27.2	30	30.7	30	30.7	34	34.9	38.7	43.6
6				21.3	25	26.4	28.6	29.4	32.5	33.2	32.5	33.2	37	37.9	42.1	47.5
7				22.4	26.4	28	30.3	31.3	34.7	35.5	34.7	35.5	39.6	40.6	45.2	51
7.5				22.9	27.1	28.7	31.2	32.2	35.7	36.5	35.7	36.5	40.8	41.8	46.6	52.6
8					27.7	29.4	32	33	36.7	37.5	36.6	37.5	41.9	43	48	54.3
9											38.5	39.3	43.1	45.3	50.5	57.2
10											40	41	46	47.3	52.9	60
11											41.4	42.5	47.8	49.2	55.1	62.6
12											42.7	43.8	49.5	50.9	57.1	65
13											43.8	45	51	52.5	59	67.3
14													52.4	53.9	60.8	69.4

* The relationship between L1max and ae is given by the following formula: $L1max = 2\times\sqrt{D\times ae - ae^2}$

Fig. 56

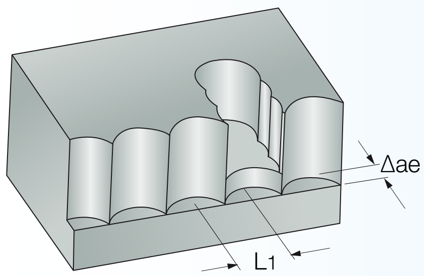


Table 88 **TANGPLUNGE HTP ...LN Plungers: Basic Cutting Speed Vo, m/min***

ISO Class DIN/ISO 513	ISCAR Mat. Group**	Vo for Grades					
		IC808	IC810	IC830	IC928	IC330	IC328
P	1	190	175	165	155	150	145
	2-4	175	155	150	140	135	135
	5	155	145	140	135	130	130
	6, 7	150	140	135	130	125	120
	8, 9	150	140	135	130	125	120
	10	140	140	135	130	125	120
	11	130	125	125	120	115	110
M	12, 13	145		135	130	125	120
K	15-16	210	230	210	200		
	17-18	175	180	170	165		
H	38.1	105		95	90		

* For 20 min. tool life
** ISCAR material group in accordance with VDI 3323 standard
☐ - More suitable grade

Plunge in chips

In plunge milling pockets and cavities, especially those with considerable depth, chip evacuation is highly important. Horizontal machines have good conditions for removing chips, however the situation with vertical machines is worse. For the latter it is very important to use the machining strategy with a tool route that ensures collecting the chips away from the tool cutting movement (at the bottom of a cavity, for example) in order to prevent compressing and re-cutting the chips, which can cause serious problems and thus significantly reduce tool life.

Table 89 **TANGPLUNGE HTP ...LN Plungers: Basic Starting Feed fzo, mm/tooth**

ISO Class DIN/ISO 513	ISCAR Material Group*	fzo for Grades			
		IC808	IC810	IC830 IC928	IC330 IC328
P	1	0.12	0.13	0.14	0.15
	2-4	0.11	0.11	0.12	0.13
	5	0.1	0.11	0.12	0.12
	6, 7	0.1	0.11	0.12	0.12
	8, 9	0.09	0.1	0.11	0.12
	10	0.08	0.09	0.1	0.11
	11	0.08	0.09	0.1	0.1
M	12, 13	0.08		0.1	0.1
K	15-16	0.13	0.15	0.16	
	17-18	0.11	0.12	0.13	
H	38.1	0.07		0.08	

* ISCAR material group in accordance with VDI 3323 standard

Please note

Plunging can be a good solution for unstable and low-power milling machine tools.

There two kinds of inserts intended for the HTP plungers: general-duty HTP LNHT…ER and HTP LNHT…ETR – that are recommended for machining hardened and hard-to-cut steel and for heavy cutting conditions.

Example

Side plunging by shell milling cutter HTP D040-4-16-R-LN10 with inserts HTP LNHT 1006 ER IC830 is proposed for machining a die cavity on a new machine tool. The cavity material is pre-hardened steel AISI P20, HRC 30…35. For necessary plunging depth 100 mm the cutter will be mounted in shell endmill holder DIN69871 40 SEMC 16X100; and this tool assembly will have an overhang of 140 mm. The width of plunge and the stepover are 5 mm and 20 mm, respectively.
Estimate starting cutting data for 60 min. tool life.

The material in the mentioned conditions represents ISCAR material group No.9.
Basic cutting speed Vo = 135 m/min (Table 88).
Basic starting feed fzo = 0.11 mm/tooth (Table 89).
Overhang/cutter diameter = 140/40 < 4, then overhang coefficient KH = 1 (Table 86).

Therefore:
Starting speed for estimated tool life 60 min. (Kt=0.8, Table 8) Vc = 135×1×0.8=108 (m/min) (may be accepted as 110 m/min)
Starting feed = 0.11×1=0.11 (mm/tooth)

Spindle speed = 1000×108/(π×40)≈860 (rpm)
Feed speed fz= 0.11×4×860≈378 (mm/min)

ITA – Your helper for finding the most efficient cutting tool

ITA, Iscar Tool Advisor, is a parametric search engine that takes computer-aided tool and process selection to an entirely new level. It "thinks" like a process engineer and allows finding the right tool resting on a huge knowledge base of best practice worldwide.
Three key features separate ITA from the other available tooling search engines:

- *Quick Search/Advanced Search Option – With just three application facts, the Quick Search provides the best starting ideas and processing strategies, with priority by productivity.*
  The Advanced Search Option takes more comprehensive application input, helps the user narrow choices down and delivers a fully developed optimum solution.
- *Accommodates Shop Preferences and Variables – The user can stipulate such preferences as solid vs. indexable tooling, tool brand/family as well as productivity/tool cost and power management priorities. Variables include machine power and condition, type of cut, available spindle speed, part shape/rigidity, setup stability, overhang, secondary operations and many more. The Variables feature makes ITA valuable for job shops as well as dedicated applications. ISCAR recognizes that most shops want to be able to use a tool for more than one job.*
- *Processing Solutions, Not Just Tooling Solutions – With Advanced Search, the user can get recommended machining parameters for a given shop environment, as well as a short list of tooling solutions.*

Visit www.iscar.com/ITA and see the ITA's true value!

Hole Making Tools

When speaking about cutting tools for die and mold making, we first of all mean milling tools. Indeed, the milling tools are so specific for die and mold manufacturing and most machining operations during this process relate to milling. However, it is hard to imagine die and mold production without hole making. Hence, we consider it necessary to add a short profile of some popular ISCAR tools for hole making.

ISCAR's broad hole making line includes a rich variety of drills, reamers, counterbores and countersinks of various kinds: solid, indexable or with interchangeable heads.

One of the most effective products for short-to-medium depth of drilling is **SUMOCHAM** – a relatively new family of drills with interchangeable carbide heads (Fig. 57). The heads are available in four cutting geometries intended for main types of engineering materials. Being clamped into steel shanks of different lengths produces efficient tools for drilling holes of various depths: from one and a half to eight hole diameters. The unique clamping mechanism allows for machining with exceptionally high cutting parameters. The shanks are designed with twisted nozzles for coolant and exhibit durable and stably constructed bodies with excellent chip evacuation properties.

SUMODRILL is a family of relatively large diameter drills (more than 60 mm). It features a drill body that carries cartridges with square indexable inserts. Each drill is supplied with a set of a shim plates for adjusting the drill's diameter within a corresponding range if it will be necessary (Fig. 58).

CHAMGUN, a family of deep hole drills with replaceable heads (Fig. 59), is intended for productive high-precision machining with good surface quality. A single drill body can carry different types of drill head profiles designed for various workpiece materials. There is no need to remove the drill for head replacement – it can be done inside the machine.

In comparison with conventional reaming methods, **BAYO T-REAM**, a family of high speed reamers with interchangeable carbide heads, allows for considerable increase of productivity. The reamers with their unique quick-change bayonet mechanism for securing the heads are suitable for reaming applications of H7 hole tolerance range. Internal coolant channels provide coolant supply directly to cutting edges.

Table 90 contains general data regarding the most often used drilling families

Fig. 57

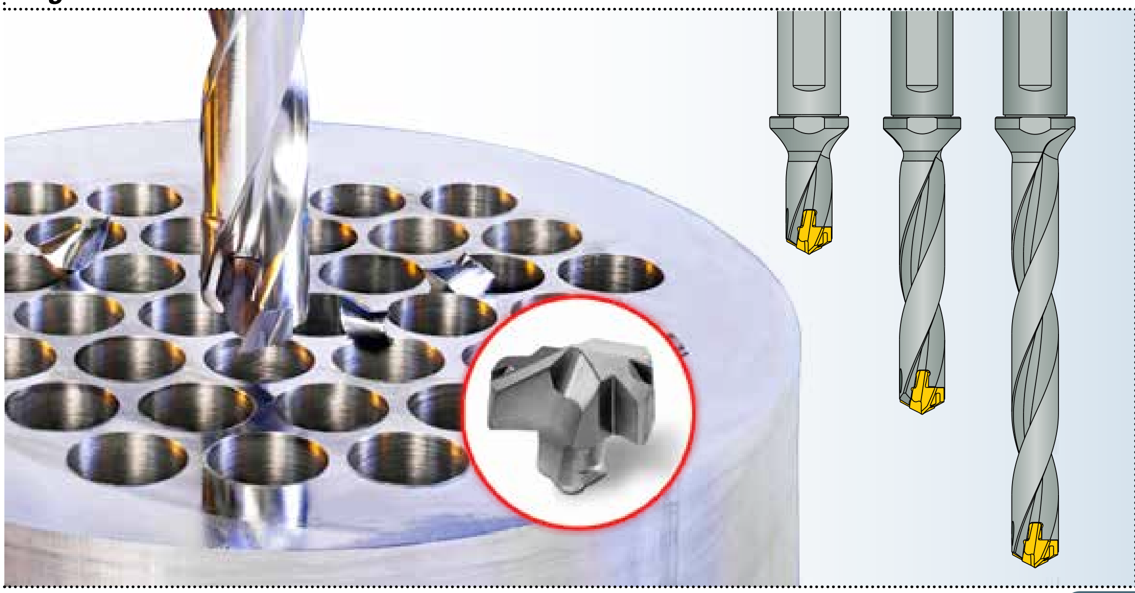


Fig. 58

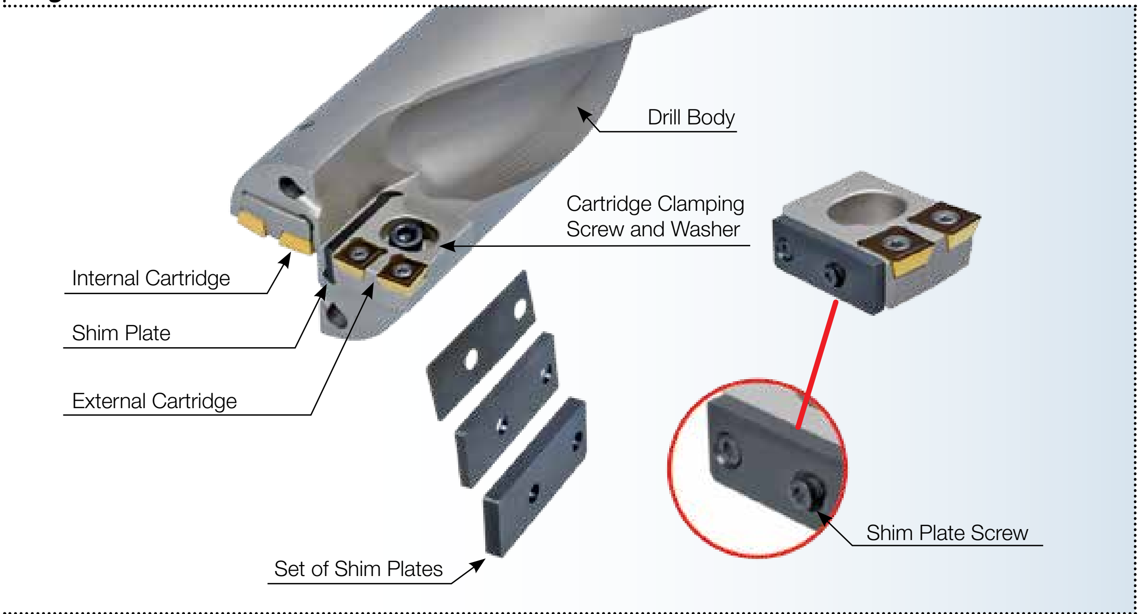


Fig. 59

Modularity and versatility

*One body that carries various heads and one head suitable for different bodies – similar to the milling cutters – such a principle is successfully applied to the hole making line. Versatile robust drills and reamers with interchangeable cutting heads designed in accordance with this principle, such as **SUMOCHAM**, **CHAMGUN** and **BAYO T-REAM**, give the die and mold maker effective tools with no setup time for increasing productivity and profitability in die and mold manufacturing.*

Table 90 **Selected ISCAR Drilling Families**

Family	Designation	Type	Diam. Range, mm	Max. Drilling Depth	Remarks
SOLIDDRILL	SCD…	Solid carbide	0.8…20	5×D*	two-flute
			3…10	8×D	
			5…10	20×D	
	SCCD…		3…20	5×D	three-flute
SUMOCHAM	DCN…	With interchangeable heads	6…25.9	12×D	
DR-TWIST	DR…	With indexable inserts	12…60	5×D	
ISCAR DR-DH	DR-DH…	With indexable inserts	25.4…69.5	5×D and up	semi-standard
SUMODRILL	DR…CA-N	With indexable inserts in cartridges	61…80	~2.5×D	adjustable
CHAMGUN	STGT…	With interchangeable heads	10…16	deep hole drilling	semi-standard

* D - drill nominal diameter

In Summation

We hope that ISCAR's reference guide will provide you with brief but necessary information, that will help you to select the most suitable cutting tool and define relevant initial cutting data.

We consider the guide to be a practical supplement to our catalogs and technical papers, and will be pleased if you will find it useful for solving real tooling problems.

The scope of the guide has allowed us only to touch upon some themes related to the cutting tools used for die and mold making. Therefore we are waiting for your response. Do you find the guide to be useful? What do you want to add? Delete? What is you own experience with ISCAR tools in die and mold manufacturing?

We are waiting for your comments in order to improve the guide and change it in cooperation with you, in order to become more effective.

3P (3): Productivity, performance, and profitability

Triple P not only emphasizes the newest products, but reflects 3 main principles, "3 pillars" of ISCAR's philosophy regarding the aim of cutting tools for the customer. By advanced tools to higher productivity for improving the work performance and to increase profitability – this is the real way for fruitful collaboration and partnership between ISCAR and the customer.

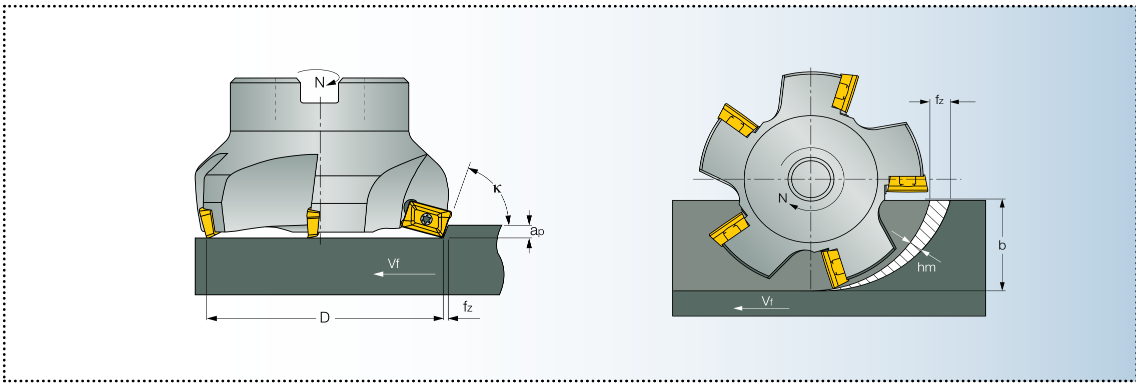
Values for Motor Power P_M

Motor Power P_M has a value of approximately 7 to 12% Maximum Machine Power

Max. Machine Power (kW)	Motor Power P_M (kW)
5.5	0.4
7.5	0.4-0.6
11.0	1.0
15.0	1.5
18.0	2.2
22.0	2.5

$P = P_c + P_M$

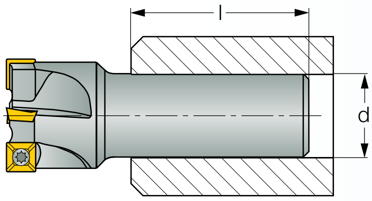
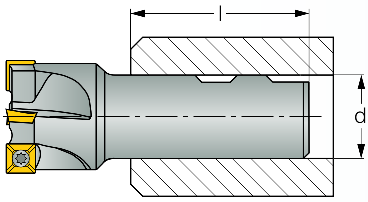
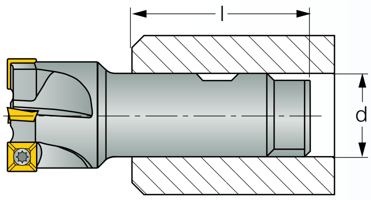
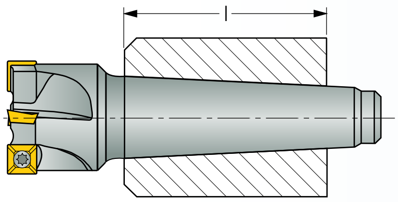
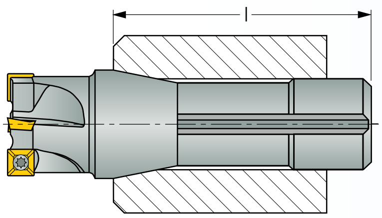
P - Total Machining Power
P_c - Net Power=Cutting Power
P_M - Motor Power (while not cutting)



Calculations	
Cutting speed	$Vc = \frac{\pi * D * N}{1000min}$ [m/min]
Spindle speed	$N = \frac{Vc * 1000}{\pi * D}$ [rev/min]
Feed speed	$V_f = f_z * Z * N$ [mm/min]
Feed per tooth	$f_z = \frac{V_f}{N * Z}$ [mm/tooth]
Feed per revolution	$f_N = fz * Z$ [mm/rev]
Metal removal rate	$Q = \frac{a_p * b * V_f}{1000}$ [cm³/min]
Machining time	$Th = \frac{L_w}{V_f}$ [min]
Specific cutting force	$K_c = K_{c1} * h_m^{-mc}$
Average chip thickness in shoulder milling for b/D≤0.1	$h_m \approx f_z * \sqrt{\frac{b}{D}}$ [mm]
Average chip thickness in shoulder milling for b/D>0.1	$hm = \frac{(\sin\kappa * 180 * b * f_z)}{\pi * D * \arcsin(b/D)}$ [mm]
Machining power	$P = \frac{(a_p * b * V_f * k_c)}{6 * 10^7 * h}$ [kW]

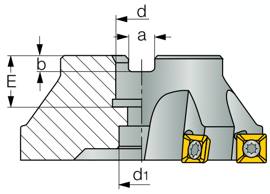
Vc	[m/min]	Cutting speed
D	[mm]	Tool diameter
N	[RPM]	Spindle speed
Vf	[mm/min]	Feed speed
fz	[mm/tooth]	Feed per tooth
Z		Number of cutting edges
f_N	[mm/rev]	Feed per revolution
Q	[cm³/min]	Metal removal rate
ap	[mm]	Depth of cut
b	[mm]	Width of cut
Th	[min]	Machining time
Lw	[mm]	Machining length
Kc	[N/mm²]	Specific cutting force
Kc_1(1)	[N/mm²]	Specific cutting force for 1 mm² chip section
Hm	[mm]	Average chip thickness
mc		Chip thickness factor
k	[degrees]	Cutting edge angle
P	[kW]	Machining power
h		Machine efficiency

Endmill Shank Styles

		Shank Diameter (d)	Recommended Min. l Value
	**Cylindrical**	10 16 20 25 32 40	1.5xd 1.5xd 1.5xd 1.5xd 1.5xd 1.5xd
	**Weldon**	12 16 20 25 32 40 50	45 48 50 56 60 70 80
	**Combined Shank (Clarkson)**	16 20 25 32 40	39 — 53 54 75
	**Morse**	CM 2 CM 3 CM 4	64 81 102.5
	**Bridgeport**		101.6

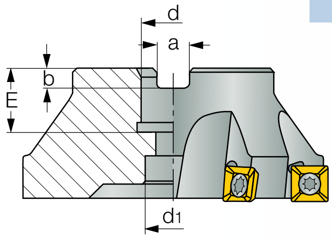
Face Mills Arbor Hole Styles

Style A



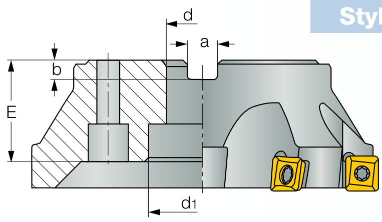
d	E	d_1	a	b
16	19	13.5	8.4	5.6
22	20	18	10.4	6.5
27	23	38	12.4	7.0

Style B



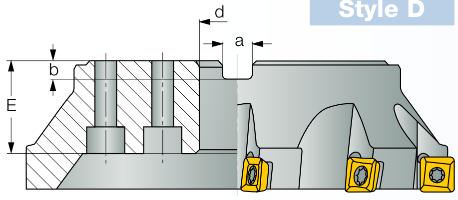
d	E	d_1	a	b
22	20	31	10.9	6.5
27	25	38	12.4	7.0
32	25	46	14.4	8.0
40	33	56	16.4	9.0

Style C



d	E	d_1	d_2	d_3	a	b
40	33	65	66.7	–	16.4	9.0
60	38	–	–	–	25.7	14.0

Style D



d	E	d_1	d_2	d_3	a	b
60	38	–	101.6	177.8	25.7	14.0

MATERIAL GROUPS

According to DIN / ISO 513 and VDI 3323

ISO	Material		Condition	Tensile Strength [N/mm²]	Kc [N/mm²]	m_c	Hardness HB	Material No.
P	Non-alloy steel and cast steel, free cutting steel	< 0.25 %C	Annealed	420	1350	0.21	125	1
P		>= 0.25 %C	Annealed	650	1500	0.22	190	2
P		< 0.55 %C	Quenched and tempered	850	1675	0.24	250	3
P		>= 0.55 %C	Annealed	750	1700	0.24	220	4
P			Quenched and tempered	1000	1900	0.24	300	5
P	Low alloy steel and cast steel (less than 5% of alloying elements		Annealed	600	1775	0.24	200	6
P				930	1675	0.24	275	7
P			Quenched and tempered	1000	1725	0.24	300	8
P				1200	1800	0.24	350	9
P	High alloy steel, cast steel, and tool steel		Annealed	680	2450	0.23	200	10
P			Quenched and tempered	1100	2500	0.23	325	11
M	Stainless steel and cast steel		Ferritic/martensitic	680	1875	0.21	200	12
M			Martensitic	820	1875	0.21	240	13
M			Austenitic	600	2150	0.20	180	14
K	Grey cast iron (GG)		Ferritic		1150	0.20	180	15
K			Pearlitic		1350	0.28	260	16
K	Cast iron nodular (GGG)		Pearlitic/ferritic		1225	0.25	160	17
K			Pearlitic/martensitic		1350	0.28	250	18
K	Malleable cast iron		Ferritic		1225	0.25	130	19
K			Pearlitic		1420	0.3	230	20
N	Aluminum-wrought alloy		Not cureable		700	0.25	60	21
N			Cured		800	0.25	100	22
N	Aluminum-cast, alloyed	<=12% Si	Not cureable		700	0.25	75	23
N			Cured		700	0.25	90	24
N		>12% Si	High temperature		750	0.25	130	25
N	Copper alloys	>1% Pb	Free cutting		700	0.27	110	26
N			Brass		700	0.27	90	27
N			Electrolytic copper		700	0.27	100	28
N	Non-metallic		Duroplastics, fiber plastics					29
N			Hard rubber					30
S	High temp. alloys	Fe based	Annealed		2600	0.24	200	31
S			Cured		3100	0.24	280	32
S		Ni or Co based	Annealed		3300	0.24	250	33
S			Cured		3300	0.24	350	34
S			Cast		3300	0.24	320	35
S	Titanium and Ti alloys			RM 400	1700	0.23		36
S			Alpha+beta alloys cured	RM 1050	2110	0.22		37
H	Hardened steel		Hardened		4600		55 HRC	38
H			Hardened		4700		60 HRC	39
H	Chilled cast iron		Cast		4600		400	40
H	Cast iron		Hardened		4500		55 HRC	41

Steel | Stainless Steel | Cast Iron
Nonferrous | High Temp. Alloys | Hardened Steel

Specific cutting force for 1 mm² chip section.
Chip thickness factor.

ISCAR MATERIAL GROUPS

According to VDI 3323 Standard

Material Group	AISI/SAE	Material No.	DIN	BS	EN	AFNOR
1	A 366 (1012) 1008	0.0030	C10	040 A 10 045 M 10 1449 10 CS		AF 34 C 10 XC 10
1		1.0028	Ust 34-2 (S250G1T)			A 34-2
1		1.0034	RSt 34-2 (S250G2T)	1449 34/20 HR, HS,CR,CS		A 34-2 NE
1		1.0035	St185 (Fe 310-0) St 33	Fe 310-0 1449 15 HR,HS		A 33
1	A 570 Gr. 33,36	1.0036	S235JRG1 (Fe 360 B) Ust 37-2	Fe 360 B 4360-40 B		
1		1.0037	S235JR (Fe 360 B) St 37-2	Fe 360 B 4360-40 B		E 24-2
1	1115	1.0038	GS-CK16	030A04	1A	
1	A 570 Gr. 40	1.0044	S275JR (Fe 430 B) St44-2	Fe 430 B FN 1449 43/25 HR, HS 4360-43 B		E 28-2
1		1.0045	S355JR	4360-50 B		E 36-2
1	A 570 Gr.50 A572 Gr.50	1.0050	E295 (Fe 490-2) St 50-2	Fe 490-2 FN 4360-50 B		A 50-2
1	A 572 Gr. 65	1.0060	E335 (Fe 590-2) St 60-2	Fe 60-2 4360-55 E; 55 C		A 60-2
1		1.0060	St60-2			
1		1.0070	E360 (Fe 690-2) St 70-2	Fe 690-2 FN		A 70-2
1		1.0112	P235S	1501-164-360B LT20		A37AP
1		1.0114	S235JU;St 37-3 U	4360-40C		E 24-3
1	A 284 Gr.D A 573 Gr.58 A 570 Gr 36;C A 611 Gr. C	1.0116	S235J2G3 (Fe 360 D 1) St 37-3	Fe 360 D1 FF 1449 37/23 CR 4360-40 D		E 24-3 E 24-4
1		1.0130	P265S	1501-164-400B LT 20		A 42 AP
1		1.0143	S275J0; St 44-3 U	4360-43C		E 28-3
1	A 573 Gr. 70 A 611 Gr.D	1.0144	S275J2G3 (Fe 430 D 1) St 44-3	Fe 430 D1 FF 4360-43 C; 43 D		E 28-3 E 28-4
1		1.0149	S275JOH; RoSt 44-2	4360-43C		
1		1.0226	DX51D; St 02 Z	Z2		GC
1	M 1010	1.0301	C10	040 A 10 045 M 10 1449 10 CS		AF 34 C 10 XC 10
1	A 621 (1008)	1.0330	DC 01 St 2; St 12	1449 4 CR 1449 3 CS		TC
1	A 619 (1008)	1.0333	Ust 3 (DC03G1) Ust 13	1449 2 CR;3 CR		E
1	A 621 (1008)	1.0334	UStW 23 (DD12G1)			S C

SS	UNI	UNE	JIS	GOST
	C 10 1 C 10	F.1511 F.151.A	S 10C	10
	Fe 330,Fe 330 B FU		SS 330	
	Fe 330 B FN			St2sp
1300	Fe 320	Fe 310-0		St0
1311 1312	FE37BFU	AE 235 B Fe 360 B		16D, 18Kp St3Kp
1311	Fe 360 B 1449 37/23 HR	AE 235 B Fe 360 B	STKM 12 A;C	
1325				
1412	Fe 430 B Fe 430 B FN	AE 275 B Fe 430 B FN	SM 400 A;B;C	St3ps; sp
2172	Fe 510 B	AE 355 B		
1550 2172	Fe 490	a 490-2 Fe 490-2 FN	SS 490	St5ps; sp
1650	Fe 60-2 Fe 590	A 590-2 Fe 590-2 FN	SM 570	St6ps; sp
	FE60-2			
1655	Fe 70-2 Fe 690	A 690-2 Fe 690-2 FN		
	Fe 360 C	AE 235 C		
	Fe 360 C	AE 235 C		
1312 1313	Fe 360 D1 FF Fe 360 C FN Fe 360 D FF Fe 37-2	AE 235 D Fe 360 D1 FF		St3kp; ps; sp 16D
		SPH 265		
1414-01	Fe 430 D	AE 275 D		
1411, 1412 1414	Fe 430 B, Fe 430 C (FN) Fe 430 D (FF)	AE 275 D Fe 430 D1 FF	SM 400 A;B;C	St4kp; ps; sp
1412-04	Fe 430 C	Fe 430 C		
1151 10	FeP 02 G	FeP 02 G		
	C 10 1 C 10	F.1511 F.151.A	S 10C	10
1142	FeP 00 FeP 01	AP 11	SPHD	15 kp
	FeP 02	AP 02	SPCD	
	FeP 12	AP 12	SPHE	10kp

ISCAR MATERIAL GROUPS

According to VDI 3323 Standard

Material Group	AISI/SAE	Material No.	DIN	BS	EN	AFNOR
1	A 622 (1008)	1.0335	DD13; StW 24	1449 1 HR		3 C
1	A 620 (1008)	1.0338	DC04 St4; St 14	1449 1 CR;2 CR		ES
1	A 516 Gr. 65; 55 A 515 Gr. 65;55 A 414 Gr. C A 442 Gr.55	1.0345	P235GH H I	1501 Gr. 141-360 1501 Gr. 161-360; 151-360 1501 Gr. 161-400; 154-360 1501 Gr. 164-360; 161-360		A 37 CP;AP
1	(M) 1020 M 1023	1.0402	C22	055 M 15, 070 M 20 1499 22 HS, CS	2C/2D	AF 42 C 20; XC 25;1 C 22
1	1020	1.0402	C22	050A20	2C/2D	CC20
1	1020;1023	1.0402	C22	055 M 15;070 M 20	2C	AF 42 C 20; XC 25;1 C 22
1		1.0425	P265GH H II	1501 Gr. 161-400;151-400 1501 Gr. 164-360; 161-400 1501 Gr. 164-400;154-400		A 42 CP; AP
1	A27 65-35	1.0443	GS-45	A1		E 23-45 M
1		1.0539	S355NH;StE 335			TSE 355-4
1		1.0545	S355N; StE 355	4360-50E		E 355 R
1		1.0546	S355NL;TStE 355	4360-50EE		E 355 FP
1		1.0547	S355JOH	4360-50C		TSE 355-3
1		1.0549	S355 NLH;TStE 355			
1		1.0553	S355JO;St 52-3U	4360-50C		E 36-3
1	A 633 Gr.C A 588	1.0562	P355N StE 355	1501 Gr.225-490A LT 20		FeE 355 KG N E 355 R/FP; A 510 AP
1		1.0565	P355NH; WStE 355	1501-225-490B LT 20		A 510 AP
1		1.0566	P355NL1; TStE 355	1501-225-490A LT 50		A 510 FP
1	1	1.0570	S355J2G3 St 52-3	Fe 510 D1 FF 1449 50/35 HR>HS 4360-50 D		E 36-3 E 36-4
1	1213	1.0715	9 SMn 28 (1SMn30)	230 M 07		S 250
1	1213	1.0715	9 SMn 28	230 M 07		S 250
1	12 L 13	1.0718	9 SMnPb 28 (11SMnPb30)			S 250 Pb
1	1108 1109	1.0721	10 S 20	(210 M 15)		10S20 10F 2
1	11 L 08	1.0722	10 SPb 20			10PbF 2
1	11 L 08	1.0722	10 SPb 20			10 PbF 2
1	1215	1.0736	9 SMn 36 11SMn37)			S 300
1	12 L 14	1.0737	9 SMnPb 36 (11SMnPb37)			S 300 Pb
1		1.0972	S315MC; QStE 300 TM	1501-40F30		E 315 D
1		1.0976	S355MC; QStE 360 TM	1501-43F35		E 355 D
1		1.0982	S460MC; QStE 460 TM	1501-50F45		

SS	UNI	UNE	JIS	GOST
	FeP 13	AP 13	SPHE	08kp
1147	FeP 04	AP 04	SPCE	08Ju; JuA
1331 1330	FeE235, Fe 360 1 KW;KG Fe 360 2 KW;KG	A 37 RC I RA II	SGV 410, SGV 450 SGV 48, SPV 450; SPV 480	
1450	C 20 C 21, C 25	1 C 22 F.112	S20C	20
1450	C20C21	F.112	S22 C	20
1450	C 20; C 21;C 25	1 C 22F.112	S 20 C;S 22 C	
1431 1430 1432	Fe 410 1 KW; KG; KT Fe 410 2 KW; KG	A 42 RC I A 42 RC II	SPV 315; SPV 355 SG 295; SGV 410 SGV 450; SGV 480	16K 20K
1305				
2134-04	Fe 510 B	Fe 355 KGN		
2334-01	FeE 355 KG	AE 355 KG		
2135-01	FeE 355 KT	AE 355 KT		
2172-04	Fe 510 C	Fe 510 C		
2135	Fe 510 D	FeE 355 KTM		
	Fe 510 C			
2106	FeE 355 KG;KW	AEE 355 KG;DD	SM 490 A;B;C; YA;YB	15GF
2106	FeE 355-2			
2107-01	FeE 355-3			
2132, 2133 2134, 2174	17GS 17G1S	AE 355 D Fe 510, D1 FF	SM 490 A;B;C;YA;YB	17GS 17G1S
1912	CF SMn 28	F.2111 - 11 SMn 28	SUM 22	
1912	CF 9 SMn 28	11 SMn 28	SUM 22	
1914	CF 9 SMnPb 28	F.2112-11 SMnPb 28	SUM 22 L SUM 23 L, SUM 24 L	
	CF 10 S 20	F. 2121 - 10 S 20		
	CF 10 SPb 20	F.2122-10 SPb 20		
	CF 10 SPb 20	10 SPb 20		
	CF 9 Mn 36	F.2113 - 12 SMn 35	SUM 25	
1926	CF 9 SMnPb 36	F.2114- 12 SMnPb 35		
2642	FeE 355TM			

ISCAR MATERIAL GROUPS

According to VDI 3323 Standard

Material Group	AISI/SAE	Material No.	DIN	BS	EN	AFNOR
1		1.0984	S500MC; QStE 500 TM			E 490 D
1		1.0986	S500MC; QStE 500 TM	1501 - 60F55		E 560 D
1	1010	1.1121	CK 10 (C10E)	040 A 10		XC 10
1		1.1121	St 37-1	4360 40 A		
1	1015	1.1141	CK 15 (C15E)	040 A 15 080 M 15	32C	XC 12 XC 15 XC 18
1	1020 1023	1.1151	C22E CK 22	055 M 15 (070 M 20)		2 C 22 XC 18 XC 25
1	D 3	1.2080	X 210 Cr 12	BD 3		Z 200 C 12
1	A36		St 44-2	4360 43 A		NFA 35-501 E 28
1			StE 320-3Z	1 501 160		
1	A572-60	1.8900	StE 380	4360 55 E		
2	(M) 1025	1.0406	C 25	070 M 26		1 C 25
2		1.0416	GS-38			20-400 M
2	A 537 Cl.1 A 414 Gr. G A 612	1.0473	P355GH	19 Mn 6		A 52 CP
2	1035	1.0501	C35	080 A 32, 080 A 35 080 M 36, 1449 40 CS		1 C 35 AF 55 C 35 XC 38
2	1045	1.0503	CF 45 (C45G)	060 A 47 080 M 46		XC 42 H 1 TS
2	1040	1.0511	C40	080 M 40		1 C 40 AF 60 C 40
2		1.0540	C 50			
2	A27 70-36	1.0551	GS-52	A2		280-480 M
2	A148 80-40	1.0553	GS-60	A3		320-560 M
2	A738	1.0577	S355J2G4 (Fe 510 D 2)	Fe 510 D2 FF 1501 Gr.224-460 1501 Gr. 224-490		A 52 FP
2	1140	1.0726	35 S 20	212 M 36	8M	35MF 6
2	1146	1.0727	45 S 20 (46S20)			45 MF 4
2	1035 1041	1.1157	40Mn4	150 M 36	15	35 M 5 40 M 5
2	1025	1.1158	C25E CK 25	(070 M 25)		2 C 25 XC 25
2	1536	1.1166	34Mn5			
2	1330	1.1170	28Mn6	(150 M 28), (150 M 18)		20 M 5, 28 Mn 6
2	1330	1.1170	28 Mn 6	150 M 5		20 M 5
2	1330	1.1170	28Mn6		14A	20M5
2		1.1178	C30E; CK 30	080M30		XC 32
2	1035	1.1180	C35R Cm 35	080 A 35		3 C 35 XC 32

SS	UNI	UNE	JIS	GOST
2662	FeE 490 TM FeE 560 TM			
1265	C 10, 2 C 10 2 C 15	F-1510-C 10 K	S 9 CK S 10 C	08;10
1300				
1370	C 15 C 16	F.1110-C 15 K F.1511-C 16 K	S 15 S 15 CK	15
1450	C 20 C 25	F.1120-C 25 K	S 20 C, S 20 CK S 22 C	20
				Ch12
1411				
1421				
2145	FeE390KG		S25C	
	C 25 1 C 25			
1306				
2101 2102	Fe E 355-2	A 52 RC I RA II	SGV 410 SGV 450 SGV 480	
1572 1550	C 35 1 C 35	F.113	S 35 C	35
1672	C 43 C 46		S 45 C	45
	C40	1 C 40	F.114.A	
1674	C 50	1 C 50		
1505				
1606				
2107		A 52 RB II AE 355 D		
1957		F.210.G		
1973				
				40G
	C25	F.1120 - C 25 K	S 25 C S 28 C	25
		TO.B	SMn 433 H	
	C 28 Mn	28 Mn 6	SCMn 1	30G
	C28Mn		SCMn1	
	C 30	2 C 30		
1572		F.1135-C 35 K-1		

ISCAR MATERIAL GROUPS

According to VDI 3323 Standard

Material Group	AISI/SAE	Material No.	DIN	BS	EN	AFNOR
2	1035 1038	1.1181	C35E CK 35	080 A 35 (080 M 36)		2 C 35, XC 32 XC 38 H 1
2	1035	1.1181	C35E CK 35	080 A 35 (080 M 36		XC 38
2	1042	1.1191	GS- Ck 45	080 A 46		XC 45
2	1049 1050	1.1206	C50E CK 50	080 M 50		2 C 50 XC 48 H 1; XC 50 H1
2	1050 1055	1.1213	Cf 53 (C53G)	070 M 55		XC 48 H TS
2	4520	1.5423	22Mo4	1503-245-420		
3		1.0050	St50-2			
3	A 516 Gr.70 A 515 Gr. 70 A 414 Gr.F; G	1.0481	P295GH 17 Mn 4	1501 Gr. 224		a 48 Cp;AP
3	1043	1.0503	C35	060 A 47 080 M 46 1449 50 HS, CS		1 C 45 AF 65 C 45
3	1074	1.0614	C 76 D; D 75-2			XC 75
3	1086	1.0616	C 86 D; D 85-2			XC 80
3	1095	1.0618	C 92 D;D 95-2			XC 90
3	1036 1330	1.1165	30Mn5	120 M 36 (150 M 28)		35 M 5
3	1335	1.1167	36Mn5	150 M 36		40 M 5
3	1040	1.1186	C40E CK 40	060 A 40, 080 A 40 080 M 40		2 C 40 XC 42 H 1
3	1045	1.1191	C45E CK 45	080 M 46 060 A 47		2 C 45 XC 42 H 1 XC 45 XC 48 H 1
3	1049	1.1201	C45R Cm 45	080 M 46		3 C 45 XC 42 H 1 XC 48 H 1
3		1.7242	18 CrMo 4			
3	A 387 Gr. 12 Cl	1.7337	16 CrMo 4 4			
3	A 387 Gr. 12 Cl.	1.7337	16 CrMo 4 4			
3		1.7362	12 CrMo 19 5	3606-625		Z 10 CD 5.05
3	A572-60		17 MnV 6	436055 E		NFA 35-501 E 36
4	1055	1.0535	C55	070 M 55		1 C 55 AF 70 C 55
4	1060	1.0601	C60	060 A 62 1449 HS,CS	43D	1 C 60 AF 70 C 55
4	107	1.0603	C67	080 A 67 1449 70 HS		XC 65

SS	UNI	UNE	JIS	GOST
1550 1572	C 35	F.1130-C 35 K	S 35 C	35
1572	C36		S35C	
1660	C45	F-1140		45
1674	C 50			50
1674	C 53		S 50 C	50
	16 Mo 5 KG; KW	F.2602- 16 Mo 5	SB 450 M; SB 480 M	
	FE50			
	Fe 510 KG;KT;KW Fe 510-2 KG;KT;KW FeE 295	A 47 RC I RA II	SG 365, SGV 410 SGV 450 SGV 480	14G2
1672 1650	C 45 1 C 45	F.114	S 45 C	45
				75
C 85				85
		F.8211-30 Mn 5 f.8311-AM 30 Mn 5	SMn 433 H SCMn 2	27ChGSNMDTL 30GSL
2120		F. 1203-36 Mn 6 F. 8212-36 Mn 5	ssmN 438 (H) SCMn 3	35G2 35GL
	C 40		S 40 C	40
1672	C 45 C 46	F.1140-C 45 K F.1142-C48 K	S 45 C S 48 C	45
1660	C 45	F.1145-C 45K-1 F.1147C 48 K-1	S 50 C	
18 CrMo 4				
	A 18 CrMo 4 5 KW			15ChM
	A 18 CrMo 4 5 KW			
	16 CrMo 20 5			
2142				
1655	C 55 1 C 55		S 55 C	55
	C 60 1 C 60		S 58 C	60(G)
	C 67			

ISCAR MATERIAL GROUPS

According to VDI 3323 Standard

Material Group	AISI/SAE	Material No.	DIN	BS	EN	AFNOR
4	1074 1075	1.0605	C75	1449 80 HS		
4	1055	1.1203	C55E CK 55	060 A 57 070 M 55		2 C 55 XC 55 H 1
4	1055	1.1209	C55R Cm 55	070 M 55		3 C 55 XC 55 H 1
4	1060 1064	1.1221	C60E CK 60	060 A 62	43D	2 C 60 XC 60 H 1
4	1070	1.1231	Ck 67 (C67E)	060 A 67		XC 68
4	1074 1075 1078	1.1248	CK 75 (C75E)	060 A 78		XC 75
4	1086	1.1269	CK 85 (C85E)			XC 90
4	1095	1.1274	Ck 101 (C101E)			XC 100
4	W 112	1.1663	C 125 W			Y2 120
4						
5		1.0070	St70-2			
5		1.7238	49 CrMo 4			
5		1.7701	51 CrMoV 4			
6	A573-81 65	1.0116	St 37-3	4360 40 B		E 24-U
6	A515 65	1.0345	H1	1 501 161		A 37 CP
6	5120	1.0841	St 52-3	150 M 19		20 MC 5
6	9255	1.0904	55 Si 7	250A53	45	55S7
6	9254	1.0904	55 Si 7	250 A 53		55 S 7
6	9262	1.0961	60SiCr7			60SC6
6	L3	1.2067	100Cr6	BL3		Y100C6
6	L1	1.2108	90 CrSi 5			
6	L2	1.2210	115CrV3			100C3
6		1.2241	51CrV4			
6		1.2311	40 CrMnMo 7			
6	4135	1.2330	35 CrMo 4	708 A 37		34 CD 4
6		1.2419	105WCr6			105WC13
6	0 1	1.2510	100 MnCrW 4	BO1		8 MO 8
6	S1	1.2542	45 WCrV7	BS1		
6	S1	1.2550	60WCrV7			55WC20
6	L6	1.2713	55NiCrMoV6			55NCDV7
6	L6	1.2721	50NiCr13			55 NCV 6
6	O2	1.2842	90MnCrV8	BO2		90 MV8
6	E 50100	1.3501	100 Cr 2			
6	52100	1.3505	100Cr6	2 S 135 535 A 99	31	100 C 6
6		1.5024	46Si7			45 S 7; Y 46 7;46 SI 7

SS	UNI	UNE	JIS	GOST
	C 75			75
1655	C 55	F.1150-C 55 K	S 55 C	55
	C 55	F.1155-C 55K-1		
1665 1678	C 60		S 58 C	60 60G, 60GA
1770	C70			65GA 68GA , 70
774	C 75			75(A)
	C 90			85(A)
1870	C 100	F-5117	SUP 4	
2223				
	FE70-2			
	51 CrMoV 4			
1312	Fe37-3			
1330				
2172	Fe 52	F-431		
2085	55Si8	56Si7		
2090				
60SiCr8	60SiCr8			
	100Cr6			Ch
2092	105WCR 5			
	107CrV3KU			
	35 cRmO 8 KU			
2234	35CrMo4	34CrMo4	SCM435TK	
2140	10WCr6	105WCr5		ChWG
2140	10WCr6	105WCr5	SKS31	
2710	45 WCrV8 KU	45WCrSi8		5ChW25F
2710	58WCr9KU			
		F.520.S	SKT4	5ChNM
2550		f-528		
2258	100Cr6	F.1310 - 100 Cr 6	SUJ2	SchCh 15
		F. 1451 - 46 SI 7		

ISCAR MATERIAL GROUPS

According to VDI 3323 Standard

Material Group	AISI/SAE (USA)	Material No. (Germany)	DIN	BS (UK)	EN	AFNOR (France)
6	9255	1.5025	51Si7			51 S 7 51 Si 7
6	9255	1.5026	55Si7	251 a 58		55 S 7
6	9260	1.5027	60Si7	251 A 60 251 H 60		60 S 7
6	9260 H	1.5028	65Si7			60 S 7
6		1.5120	38 MnSi 4			
6	A 204 Gr.A 4017	1.5415	16Mo3 15 Mo 3	1503-243 B		15 D 3
6	4419	1.5419	20Mo4	1503-243-430		
6	A 350-LF 5	1.5622	14Ni6			16N6
6	3415	1.5732	1 NilCr10			14 NC 11
6	3310; 3314	1.5752	14NilCr14	655M13	36A	12NC15
6		1.6587	17CrNiMo6	820A16		18NCD6
6		1.6657	14NiCrMo134			
6	5015	1.7015	15Cr3	523 M 15		12 C 3
6	5132	1.7033	34Cr4	530A32	18B	32C4
6	5140	1.7035	41Cr4	530M40	18	42C4
6	5140	1.7045	42Cr41	530 A 40		42 C 4 TS
6	5115	1.7131	16MnCr5	527 M 17		16 MC 5
6		1.7139	16MnCr5			
6	5155	1.7176	55Cr3	527 A 60	48	55 C 3
6	4135; 4137	1.7220	34CrMo4	708 Aa 37		35 CD 4
6	4142	1.7223	41CrMo4			
6	4140	1.7225	42CrMo4	708 M 0		42 CD 4
6		1.7228	55NiCrMoV6G	823M30	33	
6		1.7262	15CrMo5			12 CD 4
6		1.7321	20 mOcR 4			
6	ASTM A182 F-12	1.7335	13CrMo4 4	1501-620Gr27		
6	A 182-F11;12	1.7335	13 CrMo 4 4	1 501 620 Gr. 27		15 CD 4.5
6	ASTM A 182 F.22	1.7380	10CrMo9 10	1501-622gR31; 45		
6	A182 F-22	1.7380	10 CrMo 9 10	1501-622		12 CD 9.10
6		1.7715	14MoV6 3	1503-660-440		
6	A355A	1.8509	41CrAlMo 7	905 M 39	41B	40 CAD 6.12
7	A570.36	1.0038	S235JRG2 (Fe 360 B) RSt 37-2	Fe 360 B FU 1449 27/23 CR 4360-40 B		E 24-2NE
7	3135	1.5710	36NiCr6	640A35		35NC6
7		1.5755	31 NiCr 14	653 M 31		18 NC 13
7	8620	1.6523	2 NiCrMo2	805M20	362	20 NCD 2
7	8740	1.6546	40 NiCrMo 22	311-Tyre 7		
7	4130	1.7218	25CrMo4	CDS 110		25 CD 4
7		1.7733	24 CrMoV 5 5			20 CDV 6
7		1.7755	GS-45 CrMOV 10 4			

SS	UNI	UNE	JIS	GOST
2090	48 Si 7	F.1450-50 Si 7		
	50 Si 7			
2085 2090	55 Si 7	F.1440 - 56 Si 7		55S2
	60 Si 7	F. 1441 - 60 Si 7		60S2
			50 P 7 SUP 6	
2912	16Mo3(KG;KW)	F. 2601 - 16 Mo 3		
-2512	G 20 Mo 5 G 22 Mo5		SCPH 11	
14 Ni 6 KG;KT	F.2641 - 15 Ni 6			
16NiCr11	15NiCr11	SNC415(H)		
		SNC815(H)		
	14NiCrMo13			
	14NiCrMo131			
			SCr415(H)	15Ch
	34Cr4(KB)	35Cr4	SCr430(H)	35Ch
	41Cr4	42Cr4	SCr440(H)	
2245	41Cr4	42Cr4	SCr440	
2511	16MnCr5	16MnCr5		
2127				
2253			SUP9(A)	50ChGA
2234				35ChM
	41CrMo4	42CrMo4	SNB 22-1	40ChFA
2244				
2512	653M31			
2216		12CrMo4		
2625				
	14CrMo4 5	14CrMo45		
2216		12CrMo4	SCM415(H)	12ChM; 15ChM
2218	12CrMo9,10	TU.H		
		13MoCrV6		
2940	41CrAlMo7	41CrAlMo7		
1312	Fe 360 B FN	AE 235 B FN;FU		
		Fe 360 B FN; FU		St3ps; sp
2506	20NiCrMo2	20NiCrMo2	SNCM220(H)	20ChGNM
	40NiCrMo2(KB)	40NiCrMo2	SNCM240	38ChGNM
2225	25CrMo4(KB)	55Cr3	SCM420/430	20ChM; 30ChM
	21 CrMoV 5 11			

ISCAR MATERIAL GROUPS

According to VDI 3323 Standard

Material Group	AISI/SAE	Material No.	DIN	BS	EN	AFNOR
7		1.8070	21 CrMoV 5 11			
8	4142	1.2332	47 CrMo 4	708 M 40	19A	42 CD 4
8	A128 (A)	1.3401	G-X120 Mn 12			Z 120 M 12
8	3435	1.5736	36 NiCr 10			30 NC 11
8	9840	1.6511	36CrNiMo4	816M40	110	40NCD3
8	4340	1.6582	35CrNiM 6	817 M 40	24	35 NCD 6
8		1.7361	32 CeMo12	722 M 24	40B	30 CD 12
8	6150	1.8159	50 CrV 4	735 A 50	47	50CrV4
8		1.8161	58 CrV 4			
8		1.8515	32 CrMo 12	722 M 24	40B	30 CD 12
8		1.8523	39CrMoV13 9	897M39	40C	
9		1.4882	X 50 CrMnNiNbN 21 9			Z 50 CMNNb 21.09
9	3135	1.5710	36NiCr6	640A35	111A	35NC6
9		1.5864	35 niCr 18			
9			31 NiCrMo 13 4	830 m 31		
10	A573-81	1.0144	ST 44-3	4360 43 C		E 28-3
10	A 619	1.0347	DCO3 RSt;RRSt 13	1449 3 CR 1449 2 CR		E
10	M 1015 M 1016 M 1017	1.0401	C15	080 M 15 080 M 15 1449 17 CS		AF 37 C12 XC 18
10		1.0570	ST 52-3	4360 50 B		E 36-3
10	12L13	1.0718	9SMnPb28			S250Pb
10	(12L13)	1.0718	9 SMnPb 28			S 250 Pb
10		1.0723	15 S22 15 S 20	210 A 15 210 M 15		
10		1.2083				
10	H 11	1.2343	x 38 CrMoV 5 1	BH 11		Z 38 CDV 5
10	H 13	1.2344	X 40 CrMoV 5 1	BH 13		Z 40 CDV 5
10	A 2	1.2363	X100 CrMoV 5 1	BA 2		Z 100 CDV 5
10	D 2	1.2379	X 155 CrVMo 12 1	BD2		Z 160 CDV 12
10	HNV3	1.2379	X210Cr12G	BD2		Z160CDV12
10	D 4 (D 6)	1.2436	X 210 CrW 12	BD6		Z 200 CD 12
10	H 21	1.2581	X 30 WCrV 9 3	BH 21		Z 30 WCV 9
10		1.2601	X 165 CrMoV 12			
10	H 12	1.2606	X 37 CrMoW 5 1	BH 12		Z 35 CWDV 5
10	D3	1.3343	S 6-5-2	BM2		Z200C12
10	N08028	1.4563				Z1NCDU31-27-03
10	ASTM A353	1.5662	X8Ni9	1501-509;510		
10	ASM A353	1.5662	X8Ni9	502-650		9 Ni
10	2517	1.5680	12Ni19	12Ni19		Z18N5
10	2515	1.5680	12 Ni 19			Z 18 N 5
11		1.3202	S 12-1-4-5	BT 15		

SS	UNI	UNE	JIS	GOST
	35 NiCr 9			
2244	42CrMo4	42CrMo4	SCM (440)	
2183	GX120Mn12	F. 8251-AM-X120Mn12	SCMnH 1, SCMn H 11	110G13L
	36nlcRmO4(KB)	35NiCrMo4	SUP10	40ChN2MA
2541	35NiCrMo6(KB)		SNCM 447	38Ch2N2MA
2240	30CrMo12	F.124.A		
2230	50CrV4	51CrV4		50ChGFA
2240	32CrMo12	F.124.A		
	36CrMoV12			
			SNC236	
2534		f-1270		
1412			SM 400A;B;C	St4KP; ps; sp
	Fep 02	AP 02		08JU
1350	C15 C16 1 C 15	F.111	S 15 C	
2132	Fe52BFN/Fe52CFN		SM490A;B;C;YA;YB	17G5
1914	CF9SMnPb28	11SMnPb28		
1914	CF 9 SMnPb 28	11 SMnPb 28	12 L 13	
1922		F.210.F	SUM 32	
2314				
	X 37 CrMoV 5 1 KU			4Ch5MFS
2242	X40CrMoV511KU	F-5318	SKD61	4Ch5MF1S
2260	X100CrMoV51KU	F-5227	SKD12	
2310	X165CrMoW12KU	X160CrMoW12KU		
2736				
2312	X215CrW 12 1 KU	F-5213		
	X30WCrV 9 3 KU	F-526	SKD5	3Ch2W8F
2310				
	X 35 CrMoW 05 KU	F.537		5ChNM
2715	X210Cr13KU	X210Cr12	SUH3	R6M5
2584				
	14 Ni 6 KG;KT	XBNiO9		
	X10Ni9	F-2645	SL9N60(53)	
	HS 12-1-5-5	12-1-5-5		

ISCAR MATERIAL GROUPS

According to VDI 3323 Standard

Material Group	AISI/SAE	Material No.	DIN	BS	EN	AFNOR
11		1.3207	S 10-4-3-10	BT42		Z130WKCDV
11	T 15	1.3243	S 6-5-2-5			KCV 06-05-05-04-02
11		1.3246	S 7-4-2-5			Z110 WKCDV 07-05-04
11		1.3247	S 2-10-1-8	BM 42		Z110 DKCWV 09-08-04
11	M 42	1.3249	S 2-9-2-8	BM 34		
11	T 4	1.3255	S 18-1-2-5	BT 4		Z 80 WKCV 18-05-04-0
11	M 2	1.3343	S6-5-2	BM2		Z 85 WDCV
11	M 7	1.3348	S2-9-2			Z 100 DCWV 09-04-02-
11	T 1	1.3355	S 18-0-1	BT 1		Z 80 WCV 18-4-01
11	630	1.4548				Z7CNU17-04
11	HNV 3	1.4718	X45CrSi 9 3	401S45	52	Z45CS9
11	422	1.4935	x20 CrMoWV 12 1			
12	403	1.4000	X6Cr13	403 S 17		Z 6 C 13
12		1.4001	X6Cr14			
12	(410S)	1.4001	X7 Cr 13	(403 S 7)		Z 8 C 13
12	405	1.4002	X6CrA12	405S17		Z8CA12
12	405	1.4002	X6 CrAl 13	405 S 17		Z6CA13
12	416	1.4005	X12CrS 13	416 S 21		Z11 CF 13
12	410; CA-15	1.4006	(G-)X10 Cr 13	410S21	56A	Z10 C 13
12	430	1.4016	X8Cr17	Z8C17		430S15
12	430	1.4016	X6 Cr 17	430 S 15	60	Z 8 C 17
12		1.4027	G-X20Cr14	420C29		Z20C13M
12		1.4027	G-X 20 Cr 14	420 C 29		Z 20 C 13 M
12	420	1.4028	X30 Cr 13	420 S 45		Z 30 C 13
12		1.4086	G-X120Cr29	452C11		
12	430 F	1.4104	X12CrMoS17	420 S 37		Z 10 CF 17
12	440B	1.4112	X90 CrMoV 18			
12	434	1.4113	X6CrMo 17	434 S 17		Z 8 CD 17.01
12		1.4340	G-X40CrNi27 4			
12	S31500	1.4417	X2CrNiM0Si19 5			
12	S31500	1.4417	X2 CrNoMoSi 18 5 3			
12		1.4418	X4 CrNiMo16 5			Z6CND16-04-01
12	XM 8 430 Ti 439	1.4510				Z 4 CT 17
12	430tl	1.4510	X6 CrTi 17			Z 4 CT 17
12		1.4511	X 6 CrNb 17(X 6 CrNb 17			Z 4 CNb 17
12	409	1.4512	X 6 CrTi 12 (X2CrTi12)	LW 19 409 S 19		Z 3 CT 12

SS	UNI	UNE	JIS	GOST
2723	HS 6-5-2-5	6-5-2-5	SKH55	R6M5K5
7-4-2-5	HS 7-4-2-5	M 35		
2-10-1-8	HS 2-9-1-8 2-9-2-8	M 41		
R6M5				
2722	HS 6 5 2	F-5604	SKH 51	
2782	HS 2 9 2	F-5607		
R18				
	X45CrSi8	F322	SUH1	40Ch9S2
2301	X6Cr13	F.3110	SUS403	08Ch13
		F8401		08Ch13
2301				08Ch13
	X6CrAl13			
2302	X6CrAl13			
2380	X12 CrSC13	F-3411	SUS 416	
2302	X12Cr13	F.3401	SUS410	12Ch13
2320	X8Cr17	F.3113		12Ch17
2320	X8Cr17	F3113	SUS430	12Ch17
				20Ch13L
				20Ch13L
2304				20Ch13
2383	X10CrS17	F.3117	SUS430F	
2325	X8CrMo17		SUS434	
2376				
2376				
2387				
	X 6 CrTi 17	F.3115 -X 5 CrTi 17	SUS 430 LX	08Ch17T
				08Ch17T
	X 6 CrNb 17	F.3122-X 5 CrNb 17	SUS 430 LK	
	X 6 CrTi 12		SUH 409	

ISCAR MATERIAL GROUPS

According to VDI 3323 Standard

Material Group	AISI/SAE	Material No.	DIN	BS	EN	AFNOR
12		1.4720	X20CrMo13			
12	405	1.4724	X10CrA113	403S17		Z10C13
12	430	1.4742	X10CrA118	439S15	60	Z10CAS18
12	HNV6	1.4747	X80CrNiSi20	443S65	59	Z80CSN20.02
12	446	1.4749	x18 cRn 28			
12	446	1.4762	X10CrA124			Z10CAS24
12	EV 8	1.4871	X 53 CrMnNiN 21 9	349 S 54		Z 52 CMN 21.09
12	302		x12 CrNi 18 9	302 S 31		Z 10 CN 18-09
12	429		X10 CrNi 15			
13	420	1.4021	X20Cr13	420S37		Z 20 C 13
13	420	1.4031	X40 Cr 13			Z 40 C 14
13		1.4034	X46Cr13	420 S 45		Z40 C 14
13	431	1.4057	X20CrNi172	431 S 29	57	Z 15 CN 16.02
13		1.4125	X 105 CrMo 17			Z 100 CD 17
13	CA6-NM	1.4313	G-X4 CrNi 13 4	425 C 11		Z 4 CND 13-04 M
13	630	1.4542	X 5 CrNiCuNb 17 4 (X5CrNiCuNb 16-4)			
13		1.4544		S. 524 S. 526		
13	348	1.4546	X5CrNiNb 18-10	347 S 31 2 S. 130 2 S. 143/144/145 S.525/527		
13		1.4922	x20cRmV12-1			
13		1.4923	X22 CrMoV12 1			
14	304	1.4301	X 5 CrNi 18 9	304 S 15		Z 5 CN 18.09
14	303	1.4305	X10 CrNiS 18 9	303 S 21	58M	Z 8 CNF 18-09
14	304L	1.4306	X2CrNi18 9	304S12		Z2CrNi18 10
14	304L	1.4306	X2 CrNi 18 10	304 S 11		Z 3 CN 19-11
14	CF-8	1.4308	X6 CrNi 18 9	304 C 15	58E	Z 6 CN 18-10 M
14	301	1.4310	X12CrN i17 7	301 S 21		Z 12 CN 17.07
14	304 LN	1.4311	X2 CrNiN 18 10	304 S 62		Z 2 CN18.10
14		1.4312	G-X10CrNi18 8	302C25		Z10CN18.9M
14	305	1.4312	X8 CrNi 18 12	305 s 19		
14		1.4332	X2 CrNi 18-8			
14	304	1.4350	X5CrNi18 9	304S15	58E	Z6CN18.09
14	S32304	1.4362	X2 CrNiN 23 4			Z 2 CN 23-04 AZ
14	202	1.4371	X3 CrMnNiN 188 8 7	284 S 16		Z 8 CMN 18- 08-05
14	316	1.4401	X 5 CrNiMo 17 12 2 (X4 CrNiMo 17 -12-2)	316 S 13 316 S 17 316 S 19 316 S 31 316 S 33		Z 3 CND 17 -11-01 Z 6 CND 17-11 Z 6 CND 17-11-02 Z 7 CND 17-11-02 Z 7 CND 17-12-02

SS	UNI	UNE	JIS	GOST
	X10CrA112	F.311		10Ch13SJu
	X8Cr17	F.3113	SUS430	15Ch18SJu
X80CrSiNi20	F.320B	SUH4		
2322	X16Cr26		SUH446	
	X53CrMnNiN21 9		SUH35,SUH36	55Ch20G9AN4
2330				
2303	14210			20Ch13
-2304				40Ch13
X40Cr14	F.3405	SUS420J2		
2321	X16CrNi16	F.3427	SUS431	20Ch17N2
	X 105 CrMo 17			95Ch18
2385	(G)X6CrNi304		SCS5	
	X 6 CrNiTi 18 11			08Ch 18N12T
	X 6 CrNiNb 18 11			
2317	x20cRmOnl 12 01			
2332;2333				08Ch18N10
2346	X10CrNiS18.09	F.3508	SUS303	30Ch18N11
2352	x2cRnl18 11	F.3503	SCS19	
2352	X2CrNi18 11			
2333			SUS304L	
2331	X2CrNi18 07	F.3517		
2371	X2CrNiN18 10		SUS304LN	
				10Ch18N9L
				10Ch18N9L
2332	X5CrNi18 10	F.3551	SUS304	
2327				
2347	X 5 CrNiMo 17 12	F.3534-X 5 CrNiMo 17 12 2	SUS 316	

ISCAR MATERIAL GROUPS

According to VDI 3323 Standard

Material Group	AISI/SAE	Material No.	DIN	BS EN	AFNOR
14	316L	1.4404	X2 CrNiMo 17 13 2 (X2 CrNiMo 17-12-2) GX 2 CrNiMoN 18-10	316 S 11, 316 S 13 316 S 14, 316 S 31; 316 S 42, S.537;316 C 12, T.75, S. 161	Z 2 CND 17-12 Z 2 CND 18-13 Z 3 CND 17-11-02 Z 3 CND 17-12-02 FF Z 3 CND 18-12-03 Z 3 CND 19.10 M
14	316LN	1.4406	X2 CrNiMoN 17 12 2 (X2CrNiMoN 18-10)	316 S 61 316 S 63	Z2 CND 17-12 AZ
14	CF-8M	1.4408	GX 5 CrNiMoN 7 12 2 G-X 6 CrNiMo 18 10	316 C 16 (LT 196) ANC 4 B	
14		1.4410	G-X10CrNiMo18 9		Z5CND20.12M
14	316 Ln	1.4429	X2 CrNiMo 17 -13-3	316 S 62	Z 2 CND 17-13 Az
14	316L	1.4435	X2 CrNiMo18 14 3	316 S 11;316 S 13 316 S 14;316 S 31 LW 22 LWCF 22	Z 3 CND 17-12-03 Z 3 CND 18-14-03
14	316	1.4436	X 5 CrNiMo 17 13 3 (X4CRNIMO 17-13-3	316 S 19; 316 S 31 316 S 33 LW 23 LWCF 23	Z 6 CND 18-12-03 Z 7 CND 18-12-03
14	317L	1.4438	X2 CrNiMo 18 16 4 (X2CrNiMo 18-15-4)	317 S 12	Z 2 CND 19-15-04 z 3 cnd 19-15-04
14	(s31726)	1.4439	X2 CrNiMoN 17 13 5		Z 3 CND 18-14-06 AZ
14		1.4440	X 2 CrNiMo 18 13		
14	317	1.4449	X5 CrNiMo 17 13 3	317 S 16	
14	329	1.4460	X 4 CrNiMo 27 5 2 (X3CrNiMo27-5-2)		(Z 3 CND 25-07 Az) Z 5 CND 27-05 Az
14	329	1.4460	X8CrNiMo27 5		
14		1.4462	X2CrNiMoN22 5 3	318 S 13	Z 3 CND 22-05 Az (Z 2 CND 24 -08 Az) (Z 3 CND 25-06-03 Az)
14		1.4500	G-X7NiCrMoCuNb25 20		23NCDU25.20M
14	17-7PH	1.4504		316S111	
14	443 444	1.4521	X2CrMoTi18-2		
14	UNS N 08904	1.4539	X1NiCrMoCuN25-20-5		Z 2 NCDU 25-20
14	CN-7M	1.4539	(G-)X1 NiCrMoCu 25 20 5		Z1 NCDU 25-02 M
14	321	1.4541	Z 6 CrNiTi 18-10	321 S 31 321 S 51 (1010;1105) LW 24 LWCF 24	Z 6 CNT 18-10

SS	UNI	UNE	JIS	GOST
2348	X 2 CrNiMo 17 12	F.3533 - X 2 CrNiMo		
		17 13 2		
	G-X 2 CrNiMo 19 11	F.3537 - X 2 CrNiMo		
		17 13 3	SUS 316 L	
	X 2 CrNiMoN 17 12	F.3542-X 2 CrNiMoN		
		17 12 2	SUS316LN	07Ch18N; 18N10G2S2MSL
2343		F.8414-AM-X 7		
		CrNiMo 20 10	SCS 14	
2328				
2375	X 2 CrNiMoN 17 13	F.3543-X 2 CrNiMoN 17 13 3	(SUS 316 LN	
2375	X2CrNiMoN 17 13	F.3533-X 2 CrNiMo		
		17 13 2	SUS 316 L	O3Ch17N14M3
2343	X 5 CrNiMo 117 13	F.3543-X 5 CrNiMo 17 12 2	3 SUS 316	
	X 8 cRnlmO 17 13	F.3538-X 5 CrNiMo 17 13		
2367	X2CrNiMo18 16	f.3539-x 2 cRnlmO 18 16 4	SUS317L	
	X 5 CrNiMo 18 15		SUS 317	
2324		F.3309-X 8 CrNiMo 17 12 2		
		F.3552-X 8 CrNiMo 18 16 4	SUS 329 J 1	
2324				
2377			SUS 329 J3L	
	Z8CNA17-07	X2CrNiMo1712		
2326		F.3123-X 2 CrMoTiNb 18 2	SUS 444	
2562				
2564				
2337	X 6 CrNiTi 18 11	F.3523 - X 6 CrNiTi		06Ch18N10T
		18 10	SUS 321	08Ch18N10T
				09Ch18N10T
				12Ch18N10T

ISCAR MATERIAL GROUPS

According to VDI 3323 Standard

Material Group	AISI/SAE	Material No.	DIN	BS	EN	AFNOR
14	630	1.4542	X5 CrNiCuNb 17 4 (X5 CrNiChNb 16-4)			Z 7 CNU 15-05 Z 7 CNU 17-04
14	17-4PH	1.4542				Z7CNU17-04
14	S31254	1.4547	X1 CrNiMoN 20 18 7			
14	17-4PH	1.4548				Z7CNU17-04
14	347	1.4550	X6 CrNiNb 18 10	347 S 17	58F	Z 6 CNNb 18.10
14		1.4552	G-X7CrNiNb18 9			Z4CNNb19.10M
14	17-7PH	1.4568		316S111		
14	316tTi	1.4571	X6 CrNiMoTi 17 12 2	320 S 31		Z 6 CNDT 17-12002
14	316 Ti	1.4571	x 6 CrNiMoTi 17 12 2	320 S 31	58J	Z 6 NDT 17.12
14		1.4581	G-X 5 CrNiMoNb	318 C 17		Z 4 CNDNb 18.12 M
14	318	1.4583	X 10CrNiMoNb 18 12	303 S 21		Z15CNS20.12
14		1.4585	G-X7CrNiMoCuNb18 18			
14		1.4821	X20CrNiSi25 4			Z20CNS25.04
14		1.4823	G-X40CrNiSi27 4			
14	309	1.4828	X15CrNiSi20 12	309 S 24	58C	Z15CNS20.12
14		1.4829	X 12 CrNi 22 12			
14	309S	1.4833	X6 CrNi 22 13	309 S 13		Z 15 CN 24-13
14	310 S	1.4845	X12 CrNi 25 21	310S24		Z 12 CN 25-20
14	321	1.4878	X6 CrNiTi 18 9	32 1 S 20	58B	Z 6 CNT 18-12 (B)
14	Ss30415	1.4891	X5 CrNiNb 18 10			
14	S30815	1.4893	X8 CrNiNb 11			
14	304H	1.4948	X6 CrNi 18 11	304 S 51		Z 5 CN 18-09
14	660	1.4980	X5 NiCrTi 25 15			Zz 8 nctv 25-15 b ff
14			X5 NiCrN 35 25			
14	S31753		X2 CrNiMoN 18 13 4			
14			X2 CrNiMoN 25 22 7			
15	CLASS20	0.6010	GG10			Ft10D
15	A48-20B	0.6010	GG-10			FT 10 D
15	NO 25 B	0.6015	GG 15	Grade 150		FT 15 D
15	CLASS25	0.6015	GG15	GRADE150		Ft15D
15	A48 25 B	0.6015	GG 15	Grade 150		Ft 15 D
15	A48-30B	0.6020	GG-20	Grade 220		Ft 20 D
15	NO 30 B	0.6020	GG 20	Grade 220		Ft 20 D
15	A436 Type 2	0.6660	GGL-NiCr202	L-NiCuCr202		L-NC 202
15	60-40-18	0.7040	GGG 40	SNG 420/12		FCS 400-12
15	No 20 B		GG 10			Ft 10 D
16	CLASS30	0.6020	GG20	GRADE220		Ft20D
16	CLASS45	0.6030	GG30	GRADE300		Ft30D
16	A48-45 B	0.6030		Grade 300		Ft 30 D
16	A48-50	0.6035	GG-35	GRADE 350		Ft35D
16	A48-60 B	0.6040	GG40	GRADE400		Ft 40 D
16	100/70/03	0.7070	GGG-70	SNG700/2		FGS 700-2
17		0.7033	GGG35.3			

SS	UNI	UNE	JIS	GOST
			SCS 24	
			SUS 630	
2378				
2338	X6CrNiNb18 11	F.3552	SUS347	08Ch18N12B
	Z8CNA17-07	X2CrNiMo1712		09Ch17NJu1
2350				10Ch17N13M2T
2350	X6CrNiMoTi17 12	F.3535		10Ch17N13M2T
	x15cRnlsl2 12			
	X6CrNiMoTi17 12			
		F.8414	SCS17	20Ch20N14S2
2361	X6CrNi25 20	F.331	SUH310	20Ch23N18
2337	X6CrNiTi18 11	F.3553	SUS321	
2372				
2368				
2333				
2570				
110	G10			SCh10
0110-00				SCh10
0115-00	G 15	FG 15	FC150	SCh15
115	G 15	FG 15		SCh15
01 15-00	G14	FG15		SCh15
0120-00				SCh20
120	G 20		FC200	SCh20
0523-00				
0717-02	GS 370-17	FGE 38-17	FCD400	VCh42-12
110			FC100	
120	G 20	FG 20		
130	G 30	FG 30	FC300	SCh20
01 30-00				SCh30
135	G 35	FG 35	FC350	SCh30
140				SCh40
07 37-01	GGG 70	GGG 70	FCD700	
07 17-15				

ISCAR MATERIAL GROUPS

According to VDI 3323 Standard

Material Group	AISI/SAE	Material No.	DIN	BS	EN	AFNOR
17		0.7033	GGG-35.3	350/22 L 40		FGS 370/17
17	60-40-18	0.7040	GGG-40	SNG 420/12		FGS 400-12
17	60/40/18	0.7043	GGG-40.3	370/7		FGS 370/17
17	80-55-06	0.7050	GGG50	SNG500/7		FGS 500/7
17	65-45-12	0.7050	GGG-50	SNG 500/7		FGS 500-7
17		0.7652	GGG-NiMn 13 7	S-NiMn 137		S-Mn 137
17	A43D2	0.7660	GGG-NiCr 20 2	Grade S6		S-NC 202
17			GGG 40.3	SNG 370/17		FGS 370-17
18	A48-40 B	0.6025	GG25	Grade260		Ft 25 D
18		0.7060	GGG60	SNG600/3		FGS600-3
18	80/55/06	0.7060	GGG-60	600/3		FGS 600/3
18	A48 40 B					
19		0.8055	GTW55			
19	32510	0.8135	GTS-35-10	B 340/12		MN35-10
19	A47-32510	0.8135	GTS-35-10	B 340/2		Mn 35-10
19	A220-40010	0.8145	GTS-45-06	P 440/7		Mn 450-6
19			GTS-35	B 340/12		
19				8 290/6		MN 32-8
19	32510		GTS-35	B340/12		MN 35-10
20		0.8035	GTM-35	W340/3		MB35-7
20		0.8040	GTW-40	W410/4		MB40-10
20		0.8045				
20		0.8065	GTMW-65			
20	A220-50005	0.8155	GTS-55-04	P 510/4		Mn 550-4
20	50005	0.8155	GTS-55-04	P510/4		MP 50-5
20	70003	0.8165	GTS-65-02	P 570/3		Mn 650-3
20	90001	0.8170	GTS-70-02	P 690/2		Mn 700-2
20	A220-90001	0.8170	GTS-70-02			Mn 700-2
20		0.817	GTS-70-02	IP 70-2		
20	1022 1518	1.1133	20Mn5	120 M 19		20 M 5
20	1035	1.1183	Cf 35 (C35G)	080 A 35		XC 38 H 1 TS
20	400 10		GTS-45	P440/7		
20	70003		GTS-65	P 570/3		MP 60-3
21	Al99	3.0205				
21	1000	3.0255	Al99.5	L31/34/36		A59050C
21		3.3315	AlMg1			
22		3.1325	AlCuMg 1			
22		3.1655	AlCuSiPb			
22		3.2315	AlMgSi1			
22	7050	3.4345	AlZnMgCuO,5	L 86		AZ 4 GU/9051
23		3.2381	G-AlSi10Mg			
23		3.2382	GD-AlSi10Mg			
23		3.2581	G-AlSi12			

SS	UNI	UNE	JIS	GOST
0717-15				
0717-02				VCh42-12
0717-15				VCh42-12
0727-02	GGG 50			VCh50-2
	0727-02		FCD 500	
VCh50-2				
0772-00				
0776-00				
0717-12				SCh25
125	G 25	FG 25	FC250	VCh60-2
07 32-03	GGG 60	GGG 60		
0727-03			FCD600	
		GTW 55		
810		GTS 35		KCh35-10
0815-00				KCh35-10
	0852-00	GMN 45	FCMW370	
0810-00				
814			AC4A	
08 15			FCMW330	
852		GTM 35		
	GMB40	GTM 40		
	GMB45	GTM 45		KCh55-4
		GTW 65		KCh55-4
0854-00				KCh60-3
0854-00	GMN 55		FCMP490	KCh70-2
0856-00	GMN 65		FCMP590	KCh70-2
0862-00	GMN 70		FCMP690	KCh70-2
0864-00				20G
2132	G 22 Mn 3			35
	20 Mn 7	F.1515-20 Mn 6	SMnC 420	
1572	C 36; C 38		S 35 C	35
08 52				
858			FCMP540	AD0
				D1
				AD35
				AK9
811-04				
				AK12

Technical Information

ISCAR MATERIAL GROUPS

According to VDI 3323 Standard

Material Group	AISI/SAE	Material No.	DIN	BS	EN	AFNOR
23		3.3561	G-AlMg 5			
23	ZE 41	3.5101	G-MgZn4sE1Zr1	MAG 5		
23	EZ 33	3.5103	MgSE3Zn27r1	MAG 6		G-TR3Z2
23	AZ 81	3.5812	G-MgAl8Zn1	NMAG 1		
23	AZ 91	3.5912	G-MgAl9Zn1	MAG 7		
24		2.1871	G-AlCu 4 TiMg			
24		3.1754	G-AlCu5Ni1,5			
24		3.2163	G-AlSi9Cu3			
24	4218 B	3.2371	G-AlSi 7 Mg			
24	SC64D	3.2373	G-AlSI9MGWA			A-S7G
24		3.2373	G-AlSi 9 Mg			
24	QE 22	3.5106	G-MgAg3SE2Zr1	MAG 12		
24	GD-AlSI12		G-ALMG5	LM5		A-SU12
23-24	A360.2	3.2383	G-AlSi0Mg(Cu)	LM9		
23-24	A356-72			2789;1973		NF A32-201
23-24	356.1			LM25		
23-24	A413.2		G-AlSi12	LM 6		
23-24	A413.1		G-AlSi 12 (Cu)	LM 20		
23-24	A413.0		GD-AlSi12			
23-24	A380.1		GD-AlSi8Cu3	LM24		
26	C 93200	2.1090	G-CuSn 7 5 pb			U-E 7 Z 5 pb 4
26	C 83600	2.1096	G-CuSn5ZnPb	LG 2		
26	C 83600	2.1098	G-CuSn 2 Znpb			
26	C 23000	2.1182	G-CuPb15Sn	LB1		U-pb 15 E 8
26	C 93800	2.1182	G-CuPb15Sn			Uu-PB 15e 8
27		2.0240	CuZn 15			
27	C 27200	2.0321	CuZn 37	CZ 108		CuZn 36, CuZn 37
27	C 27700	2.0321	CuZn 37	CZ 108		CuZn 36, CuZn 37
27		2.0590	G-CuZn40Fe			
27	C 86500	2.0592	G-CuZn 35 Al 1	U-Z 36 N 3		HTB 1
27	C 86200	2.0596	G-CuZn 34 Al 2	HTB 1		U-Z 36 N 3
27	C 18200	2.1293	CuCrZr	CC 102		U-Cr 0.8 Zr
28		2.0060	E-Cu57			
28		2.0375	CuZn36Pb3			
28	C 94100	2.0596	G-CuZn 34 Al 2	HTB 1		U-Z 36 N 3
28	C 63000	2.0966	CuAl 10 Ni 5 Fe 4	Ca 104		U-A 10 N
28	B-148-52	2.0975	G-CuAl 10 Ni			
28	C 90700	2.1050	G-CuSn 10	CT1		
28	C 90800	2.1052	G-CuSn 12	pb 2		UE 12 P
28	C 81500	2.1292	G-CuCrF 35	CC1-FF		
28		2.4764	CoCr20W15Ni			
31	N 08800	1.4558	X 2 NiCrAlTi 32 20	NA 15		
31	N 08031	1.4562	X 1 NiCrMoCu 32 28 7			
31	N 08028	1.4563	X 1 NiCrMoCuN 31 27 4			

Member IMC Group
ISCAR

SS	UNI	UNE	JIS	GOST
				VAL8
				AK8
4251			C4BS	AK9
4252				
4253				
4244			A5052	AK7
4261				
4260			ADC12	AK12
4247			A6061	
4250			A7075	
	C 2700			L63
	C2720			L63
				LTs23AD; ZMts
				LS60-2
				BrAD; N10-4-4
2584				EK77

ISCAR MATERIAL GROUPS

According to VDI 3323 Standard

Material Group	AISI/SAE	Material No.	DIN	BS	EN	AFNOR
31	N 08330	1.4864	X 12 NiCrSi 36 16	NA 17		Z 12 NCS 35.16
31	330	1.4864	X12 NiCrSi 36 16	NA 17		Z 12 NCS 37.18
31		1.4865	G-X40NiCrSi38 18	330 C 40		
31		1.4958	X 5 NiCrAlTi 31 20			
31	AMS 5544	LW2.4668	NiCr19NbMo			NC20K14
32		1.4977	X 40 CoCrNi 20 20			Z 42 CNKDWNb
33	Monel 400	2.4360	NiCu30Fe	NA 13		NU 30
33	5390A	2.4603				NC22FeD
33	Hastelloy C-4	2.4610	NiMo16cR16Ti			
33	Nimonic 75	2.4630	NiCr20Ti	HR 5,203-4		NC 20 T
33		2.4630	NiCr20Ti	HR5,203-4		NC20T
33	Inconel 690	2.4642	NiC29Fe			Nnc 30 Fe
33	Inconel 625	2.4856	NiCr22Mo9Nb	NA 21		NC 22 FeDNb
33	5666	2.4856	NiCr22Mo9Nb			Inconel 625
33	Incoloy 825	2.4858	NiCr21Mo	NA 16		NC 21 Fe DU
34	Monel k-500	2.4375	NiCu30 Al	NA 18		NU 30 AT
34	4676	2.4375	NiCu30Al	3072-76		
34		2.4631	NiCr20TiAl	Hr40;601		NC20TA
34	Inconel 718	2.4668	NiCr19FeNbMo			NC 19 Fe Nb
34	Inconel	2.4694	NiCr16fE7TiAl			
34		2.4955	NiFe25Cr20NbTi			
34	5383	LM2.4668	NiCr19Fe19NbMo	HR8		NC19eNB
34	5391	LW2 4670	S-NiCr13A16MoNb	3146-3		NC12AD
34	5660	LW2.4662	NiFe35Cr14MoTi			ZSNCDT42
34	5537C	LW2.4964	CoCr20W15Ni			KC20WN
34	AMS 5772		C0Cr22W14Ni			KC22WN
35	Inconel X-750	2.4669	NiCr15Fe7TiAl			NC 15 TNb A
35	Hastelloy B	2.4685	G-NiMo28			
35	Hastelloy C	2.4810	G-NiMo30			
35	AMS 5399	2.4973	NiCr19Co11MoTi			NC19KDT
35		3.7115	TiAl5Sn2			
36	R 50250	3.7025	Ti 1	2 TA 1		
36	R 52250	3.7225	Ti 1 pd	TP 1		
36	AMS 5397	LW2 4674	NiCo15Cr10MoAlTi			
37		3.7124	TiCu2	2 TA 21-24		
37	R 54620	3.7145	TiAl6Sn2Zr4Mo2Si			
37		3.7165	TiAl6V4	TA 10-13;TA 28		T-A 6 V
37		3.7185	TiAl4Mo4Sn2	TA 45-51; TA 57		
37		3.7195	TiAl 3 V 2.5			
37			TiAl4Mo4Sn4Si0.5			
37	AMS R54520		TiAl5Sn2.5	TA14/17		T-A5E
37	AMS R56400		TiAl6V4	TA10-13/TA28		T-A6V
37	AMS R56401		TiAl6V4ELI	TA11		

Member IMC Group

SS	UNI	UNE	JIS	GOST
			SUH330	
	XG50NiCr39 19		SCH15	
				KhN38VT
				KhN77TYuR
				VT5-1
				VT1-00
				VT6

Technical Information

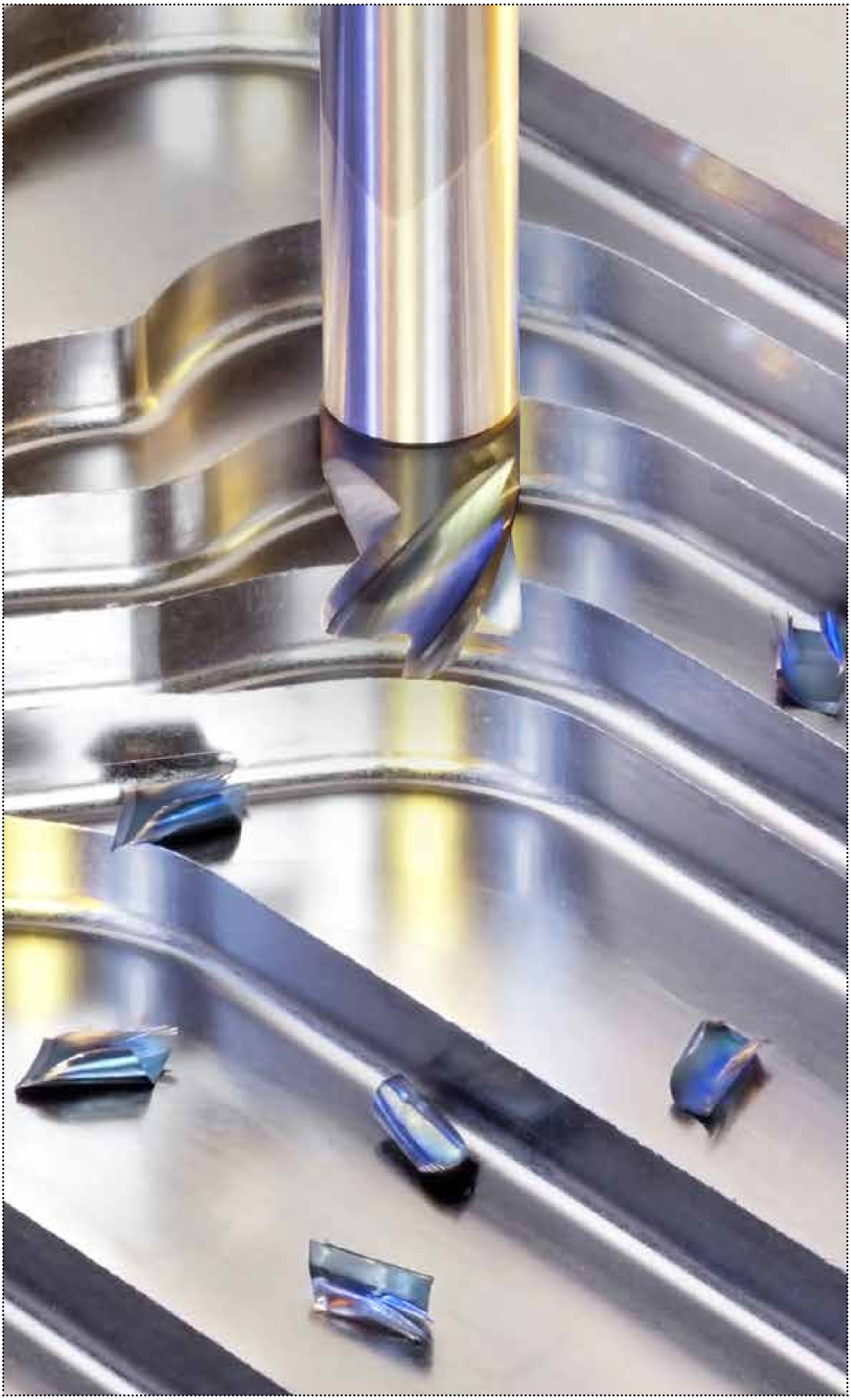
Die and Mold

ISCAR MATERIAL GROUPS

According to VDI 3323 Standard

Material Group	AISI/SAE	Material No.	DIN	BS	EN	AFNOR
38	W 1	1.1545	C 105 W1	BW 1A		Y1 105
38	W210	1.1545	C105W1	BW2		Y120
38		1.2762	75 CrMoNiW 6 7			
38	440C	1.4125	X105 CrMo 17			Z 100 CD 17
38		1.6746	32 nlcRmO 14 5	832 M 31		35 NCD 14
40	Ni- Hard 2	0.9620	G-X 260 NiCr 4 2	Grade 2 A		
40	Ni- Hard 1	0.9625	G-X 330 Ni Cr 4 2	Grade 2 B		
40	Ni-Hard 4	0.9630	G-X 300 CrNiSi 9 5 2			
40		0.9640	G-X 300 CrMoNi 15 2 1			
40	A 532 III A 25% Cr	0.9650	G-X 260 Cr 27	Grade 3 D		
40	A 532 III A 25% Cr	0.9655	G-X 300 CrNMo 27 1	Grade 3 E		
40		1.2419	105 WCr 6	105WC 13		
40	310	1.4841	X15 CrNiSi 25 20	314 S31		Z 15 CNS 25-20
41		0.9635	G-X 300 CrMo 15 3			
41		0.9645	G-X 260 CrMoNi 20 2 1			
41		0.9655	G-X 300 CrNMo 27 1			

SS	UNI	UNE	JIS	GOST
1880	C 100 KU	F-5118	SK 3	U10A
2900	C120KU	CF.515	SUP4	U10A
				95Ch18
	0512-00			
	0513-00			
	0466-00			ChWG
				20Ch25N20S2
		107 WCr 5 KU		



Member IMC Group

ISCAR LTD.
Headquarters
Tefen 24959, Israel
Tel +972 (0)4 997 0311
Fax +972 (0)4 987 3741
www.iscar.com
headquarter@iscar.co.il

Argentina
ISCAR TOOLS ARGENTINA SA
Monteagudo 222
1437 Buenos Aires
Tel +54 114 912 2200
Fax +54 114 912 4411
admin@iscararg.com.ar

Australia
Headquarters and Technical Centre
Norwest Business Park
30 Brookhollow Avenue
Baulkham Hills NSW 2153
Australian Technical Training Centre
Bell Street, Preston, Victoria, 3072
Tel +61 (0) 2 8848 3500
Fax +61 (0) 2 8848 3511
iscaraus@iscar.com.au
www.iscar.com.au

Austria
ISCAR AUSTRIA GmbH
Im Stadtgut C 2
A-4407 Steyr-Gleink
Tel +43 7252 71200-0
Fax +43 7252 71200-999
office@iscar.at
www.iscar.at

Belarus
JV ALC "TWING-M"
Slutskaya str. 3,
223056 v. Sennitsa
Minsk district
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.
Roekhout 13
B 1702 Dilbeek (Groot-Bijgaarden)
Tel +32 (0) 2 464 2020
Fax +32 (0) 2 522 5121
info@iscar.be
www.iscar.be

Bosnia
(Representative Office)
Kralja Tvrtka I br. 17
BIH- 72000 Zenica
Tel +387 32 201 100
Fax+387 32 201 101
info@iscar.ba

Brazil
ISCAR DO BRASIL COML. LTDA.
Rodovia Miguel Melhado Campos,
Km 79, Bairro Moinho
CEP: 13280-000 - Vinhedo - SP
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
Starozagorska 1, Str.
Floor 1, Office G,
6100 Kazanlak
Tel/Fax +359 431 62557
aa_iscar@infotel.bg

Canada
ISCAR TOOLS INC.
2100 Bristol Circle
Oakville, Ontario L6H 5R3
Tel +1 905 829 9000
Fax +1 905 829 9100
admin@iscar.ca
www.iscar.ca

China
ISCAR CHINA
7B21, Hanwei Plaza,
7 Guanghua Road
Chaoyang District
Bejiing 100004
Tel +86 10 6561 0261/2/3
Fax +86 10 6561 0264
iscar@iscar.com.cn
www.iscar.com.cn

Croatia
ISCAR ALATI d.o.o.
J. Jelačića 134
CRO-10430 Samobor
Tel +385 (0) 1 33 23 301
Fax +385 (0) 1 33 76 145
iscar@zg.t-com.hr
www.iscar.hr

Czech Republic
ISCAR CR s.r.o.
Mánesova 73, 301 00 Plzen
Tel +420 377 420 625
Fax +420 377 420 630
iscar@iscar.cz
www.iscar.cz

Finland
ISCAR FINLAND OY
Ahertajantie 6
02100 Espoo
Tel +358-(0)9-439 1420
Fax +358-(0)9-466 328
info@iscar.fi

France
ISCAR FRANCE SAS
8, Rue Georges Guynemer
78286 GUYANCOURT Cedex
Tel +33 (0)1 30 12 92 92
Fax +33 (0)1 30 43 88 22
info@iscar.fr
www.iscar.fr

Germany
ISCAR GERMANY GmbH
Eisenstockstrasse 14
D 76275 Ettlingen
Tel +49 (0) 72 43 9908-0
Fax +49 (0) 72 43 9908-93
gmbh@iscar.de
www.iscar.de

Hungary
ISCAR HUNGARY kft
Kassai u 151
H 1142 Budapest
Tel +36 1 251 5688
Fax +36 1 251 4757
iscar@iscar.hu
www.iscar.hu

Italy
ISCAR ITALIA srl
Via Mattei 49 / 51
20020 Arese [MI]
Tel +39 02 93 528 1
Fax +39 02 93 528 213
marketing@iscaritalia.it
www.iscaritalia.it

Japan
ISCAR JAPAN LTD.
Head Office
15th Floor, Senri Asahi
Hankyu Building
1-5-3, Shinsenri-Higashimachi
Toyonaka-Shi,
Osaka 560-0082
Tel +81 6 6835 5471
Fax +81 6 6835 5472
iscar@iscar.co.jp
www.iscar.co.jp

Macedonia
(Representative Office)
Londonska 19/4
MK-1000 Skopje
Tel +389 2 309 02 52
Fax +389 2 309 02 54
info@iscar.com.mk

Mexico
ISCAR DE MÉXICO,
S.A de C.V.
Fray Pedro de Gante 15
Col. Cimatario
Querétaro, Qro.
C.P. 76030
Tel +52 (442) 214 5505
Fax +52 (442) 214 5510
iscarmex@iscar.com.mx
www.iscar.com.mx

The Netherlands
ISCAR NEDERLAND B.V.
Postbus 704, 2800 AS Gouda
Tel +31 (0) 182 535523
Fax +31 (0) 182 572777
info@iscar.nl
www.iscar.nl

New Zealand
ISCAR PACIFIC LTD.
1/501 Mt. Wellington Hwy.
Mt. Wellington Auckland
Tel +64 9 5731280
Fax +64 9 5730781
iscar@iscarpac.co.nz

Poland
ISCAR POLAND Sp. z o.o.
ul. Gospodarcza 14
40-432 Katowice
Tel +48 32 735 7700
Fax +48 32 735 7701
iscar@iscar.pl
www.iscar.pl

Portugal
ISCAR PORTUGAL, SA
Avd.Dr.Domingos Caetano de Sousa
Fracção B, Nº 541
4520-165 Santa Maria da Feira
Tel +351 256 579950
Fax +351 256 586764
geral@iscarportugal.pt
www.iscarportugal.pt

Romania
ISCAR TOOLS SRL
Str. Maramures nr. 38,
Corp 2, Otopeni,
jud. Ilfov, cod 010832
Tel +40 (0)312 286 614
Fax +40 (0)312 286 615
iscar-romania@iscar.com

Russia
Moscow
ISCAR RUSSIA CIS
Godovikova str. 9, build. 10
129085 Moscow
Tel/fax +7 495 660 91 25/31
iscar@iscar.ru
www.iscar.ru

Chelyabinsk
ISCAR RF EAST LTD
Malogruzovaya str., 1 - office 605
454007, Chelyabinsk
Tel/fax +7 351 2450432
rfe@iscar.com
www.iscar.ru

Serbia
ISCAR TOOLS d.o.o.
Autoput 22
SRB-11080 Zemun
Tel +381 11 314 90 38
Fax +381 11 314 91 47
info@iscartools.rs

Slovakia
ISCAR SR, s.r.o.
K múzeu 3
010 03 Zilina
Tel +421 (0) 41 5074301
Fax +421 (0) 41 5074311
info@iscar.sk
www.iscar.sk

Slovenia
ISCAR SLOVENIJA d.o.o.
IOC, Motnica 14
SI-1236 Trzin
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.
47 Lake Road
Longmeadow Business Estate - North
Extension 7, Modderfontein,
Edenvale, Gauteng
P.O. Box 392
Longmeadow Business Estate - North 1609
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
304 Youggye-Ri,
Gachang-myeon
Dalsung-gun, Daegu 711-860
Tel +82 53 760 7590
Fax +82 53 767 7203
koss@taegutec.co.kr
www.iscarkorea.co.kr

Spain
ISCAR IBERICA SA
Parc Tecnològic del Vallès
Avda. Universitat Autònoma 19-21
08290 Cerdanyola-Barcelona
Tel +34 93 594 6484
Fax +34 93 582 4458
iscar@iscarib.es
www.iscarib.es

Sweden
ISCAR SVERIGE AB
Kungsangsvagen 17B
Box 845
751 08 Uppsala
Tel +46 (0) 18 66 90 60
Fax +46 (0) 18 122 920
info@iscar.se
www.iscar.se

Switzerland
ISCAR HARTMETALL AG
Wespenstrasse 14
CH 8500 Frauenfeld
Tel +41 (0) 52 728 0850
Fax +41 (0) 52 728 0855
office@iscar.ch
www.iscar.ch

Taiwan
ISCAR TAIWAN LTD.
395, Da Duen South Road,
Taichung 408
Tel +886 (0)4-24731573
Fax +886 (0)4-24731530
iscar.taiwan@msa.hinet.net
www.iscar.org.tw

Thailand
ISCAR THAILAND LTD.
57, 59, 61, 63 Soi Samanchan-Babos
Sukhumvit Rd.
Phra Khanong, Khlong Toey
Bangkok 10110
Tel +66 (2) 7136633-8
Fax +66 (2) 7136632
iscar@iscarthailand.com

Turkey
ISCAR KESICI TAKIM
TIC. VE IML. LTD. STI.
Gebze Organize Sanayi Bölgesi (GOSB)
Ihsan Dede Cad. No: 105
Gebze / Kocaeli
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
Volgodonska str., 66
02099 Kiev
Tel/fax +38 (044) 503-07-08
iscar_ua@iscar.com
www.iscar.ru

United Kingdom
ISCAR TOOLS LTD.
Woodgate Business Park
Bartley Green
Birmingham B32 3DE
Tel +44 (0) 121 422 8585
Fax +44 (0) 121 423 2789
sales@iscaruk.co.uk
www.iscaruk.co.uk

United States
ISCAR METALS INC.
300 Westway Place
Arlington, TX 76018-1021
Tel +1 817 258 3200
Tech Tel 1-877-BY-ISCAR
Fax +1 817 258 3221
info@iscarmetals.com
www.iscarmetals.com

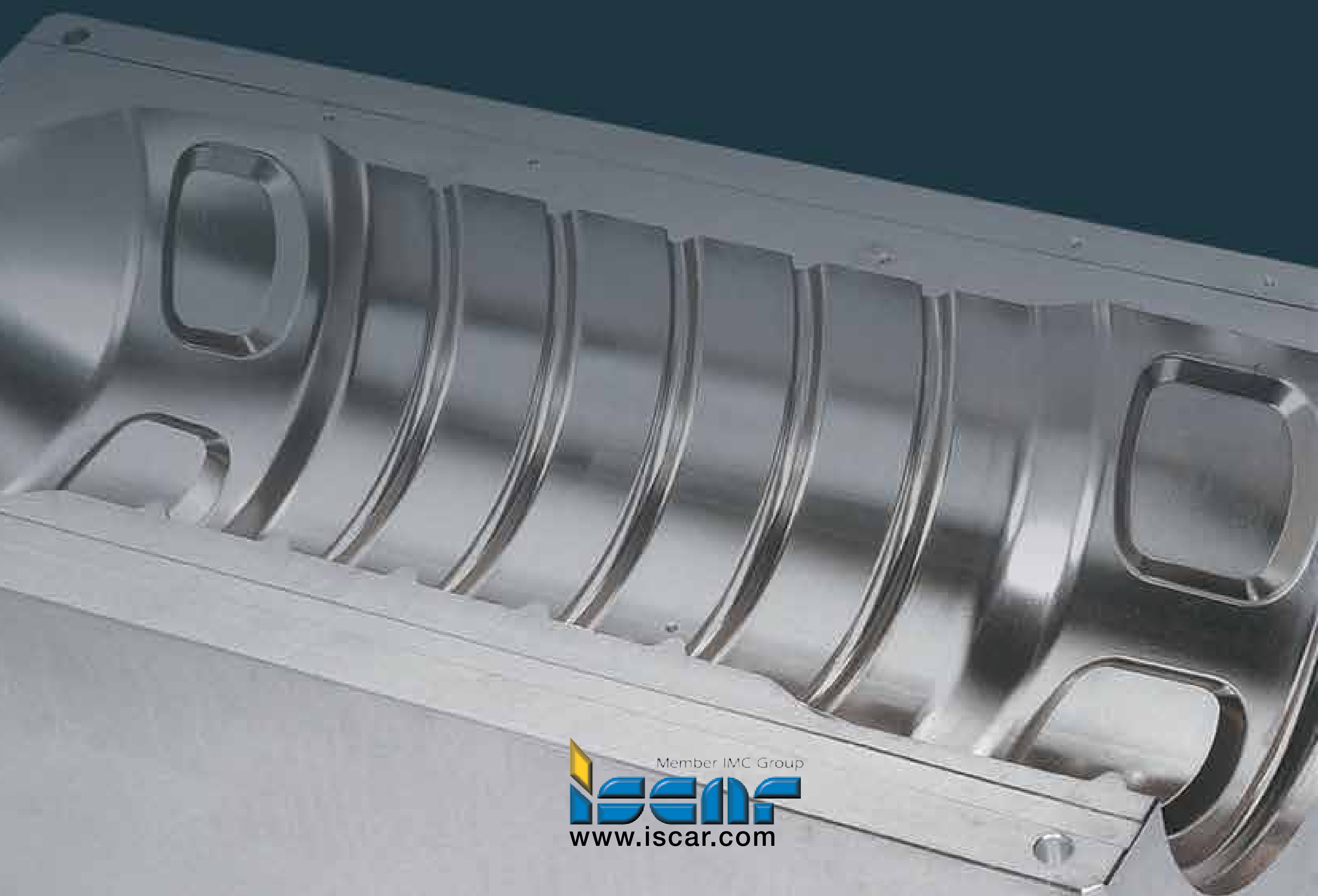
Vietnam
ISCAR VIETNAM
(Representative Office)
Room D 2.8, Etown Building,
364 Cong Hoa, Tan Binh Dist.,
Ho Chi Minh City
Tel +84 38 123 519/20
Fax +84 38 123 521
iscarvn@hcm.fpt.vn

02/2017

7861458

ISCAR's Quick Reference Guide for Die and Mold Making

Die and Mold



Member IMC Group

www.iscar.com