Technical Support

The technical section of this catalog is intended to provide sufficient basic information to apply indexable cutting tools, specifically OTM products. The majority of requests for technical support that OTM receives are for speeds and feeds. If you need technical help beyond what is contained in this section, contact our technical staff.

APPLICATION CONSIDERATIONS

Feed and Speeds

Proper cutting speeds and feeds are obviously important considerations. OTM has developed some recommended starting points through testing in our shop and in the field.

Rigid Setups

No tool will perform at maximum potential in a poor setup. Workpieces must be held securely, and tool extension should be held to a minimum. Tombstones must be mounted properly, and workholding must be adequate. Worn jaws can introduce chatter, and possibly mis-locate parts, or even release parts during the cut. The machine itself must be in good condition, and **tool gage lengths should be as short as possible**. Shorter gage lengths result in higher feed capabilities with less deflection and chatter, and longer insert life.

Machine Horsepower vs. Spindle Horsepower

There can be a great difference between the spindle motor rating (machine horsepower) and the horsepower at the tool tip (spindle horsepower). Typically, spindle horsepower is around 75% of rated horsepower.

DRILLING BASICS

Proper drilling involves proper machine setup, coolant delivery, insert selection, and a basic understanding of the drilling process. Indexable drills are not the same as twist drills in application or performance.

BASIC DRILLING

Drilling a hole from solid stock is a tough job for a variety of reasons. Chip and heat evacuation are tough, the actual cutting action is hidden from view, and the surface footage varies from the programmed speed at the outer cutting edge to zero at the center of the tool. A drill doesn't actually cut at the center, it pushes the material out of the way. Cutting forces are high and other factors such as coolant delivery and machine alignment can combine to cause a drill to deflect and cut off-size, or fail catastrophically.

MACHINE SETUP

This is an important consideration when using indexable drills. In drilling, shorter is always better. Use the shortest drill and the shortest adapter possible. This will always improve performance. The machine spindle and bearings should be in good condition, and the fixturing must be adequate and rigid.

In lathes, *alignment is crucial.* The tool must run within 0.003" TIR to the spindle, or insert failure could occur. **If the center insert continually chips, check the alignment.** Chuck jaws should be in good condition as well and provide adequate clamping pressure. The workpiece should not extend from the chuck excessively either.

COOLANT DELIVERY

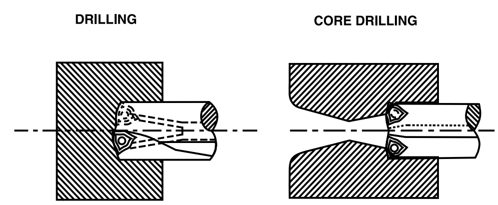
Coolant thru-the-spindle should be used wherever possible. Coolant improves tool life by lubricating the cut and removing heat and chips from the work area. Chip evacuation is very important when drilling.

INDEXABLE VS. TWIST DRILLS

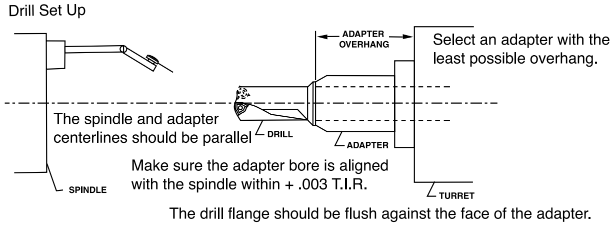
Indexable drills are very different than twist drills. Replaceable inserts eliminate drill regrinding. Because of higher surface footages, indexable drills generally out-feed twist drills.

DRILLS VS. CORE DRILLS

OTM drills are designed to put holes in solid material. OTM core drills are used to open a pre-existing hole. Core drills, because of their design, can attain two times the feed rate of insert drills. As a result, core drills provide an excellent means enlarging an existing hole.



Drill Set Up



DRILL APPLICATION HOLESHOT INDEXABLE DRILL

STATIONARY

For stationary (lathe) applications, the drill body should be mounted concentric with the spindle center line within + .003". Longitudinal alignment should be within .005" in six inches. Insert failure can result if alignment is too far above or below the centerline. Flats on the shank should be precisely aligned so that the cutting edges are parallel to the x-axis, which will help chip flow.

A disc is normally produced as the drill breaks through the hole. Although the disc is usually minimal with HOLESHOT, adequate guarding should be provided for and in place.

HOLESHOT drills can be slightly offset .004" - .006", on lathe applications, to drill other than nominal sizes from solid. Offset can only be performed for plus sizing in the axis which increases/decreases cutting diameter. Offset in the other axis changes above/below center cutting conditions, resulting in breakage. Select an adapter with the least possible overhang.

ROTATING

Make certain the spindle is rigid with minimal runout. Since both machine and fixture rigidity are key factors, make sure the workpiece is rigidly fixtured and secured. Mount drill for the least possible overhang and make sure the drill flange is flush against the face of the adapter.

Coolant - Chip removal and tool life are enhanced by feeding coolant through the drill. 30 P.S.I. minimum coolant pressure is recommended for horizontal applications. Vertical position requires a higher coolant pressure (40 to 60 P.S.I.) to flush chips properly. HOLESHOT accepts standard coolant glands.

Through the tool coolant is preferred for HOLESHOT drilling, but due to the unique flute design, coolant deficiencies can be overcome. Especially in smaller, lower horsepower machine tools, strong flood coolant can be utilized with excellent results. When flooding the cut, direct coolant directly into the drilling area.

FEED / SPEED CALCULATIONS

UNIT CONVERSIONS

Feed and speed calculations use the following abbreviations:

FPT = Feed Per Tooth (Drills have one effective tooth, coredrills have two effective teeth)
FR = Feed Rate
IPM = Inches Per Minute
IPR = Inches Per Revolution
RPM = Revolutions Per Minute
SFM = Surface Feet Per Minute

The formulas are:

FPT = (IPR x RPM) / effective teeth
IPM = IPR x RPM or FPT x effective teeth x RPM
IPR = FPT x effective teeth
RPM = 3.82 x (SFM / drill diameter)
SFM = 0.262 x drill diameter x RPM

Cutting cycle time = length of cut (inches) / IPM

SURFACE SPEED PER MINUTE

SFM = .262 x DIA x RPM

REVOLUTIONS PER MINUTE

$$RPM = \frac{3.82 \times SFM}{DIA}$$

FEEDRATE (inches/minute)

IPM = IPR x RPM

FEEDRATE (inches/revolution)

$$IPR = \frac{t_{chip}}{\cos\alpha}$$

METAL REMOVAL RATE

3 x Dia x IPR x SFM for Drilling
12 x WOC x IPR x SFM for Core Drilling

TIME IN CUT (seconds)

$$T = \frac{15.7 \times DIA \times LOC}{SFM \times IPR}$$

or

$$T = \frac{60 \times LOC}{IPM}$$

DEFINITION OF TERMS

DIA = DIAMETER OF THE DRILL (INCHES)
DOC = DEPTH OF CUT (INCHES)
EFF = MACHINE EFFICIENCY
HPM = HORSEPOWER REQUIRED AT MOTOR
HPS = HORSEPOWER REQUIRED AT SPINDLE
IPM = FEEDRATE (INCHES PER MINUTE)
IPR = FEEDRATE (INCHES PER REVOLUTION)
FR = FEEDRATE (SEE IPM AND IPR)a = LEAD ANGLE
LOC = LENGTH OF CUT (INCHES)
Q = METAL REMOVAL RATE (CU. INCHES PER MINUTE)
RPM = REVOLUTIONS PER MINUTE
SFM = SURFACE FEET PER MINUTE
T = TIME (IN SECONDS)
t_{chip} = CHIP THICKNESS (INCHES)
UHP = UNIT HORSEPOWER FACTOR

DRILLING SPEEDS AND FEEDS
HOLESHOT® DRILLS AND COREMASTER™ COREDRILLS

NOTE: Drills are one-flute effective tools. Coredrills are two flute effective tools. Feedrates should be calculated accordingly.

MACHINING RECOMMENDATIONS

Material	Condition	Insert Grade*	SFM	FPT	Comments
Aluminum	6061	6N	1800-2000	.008-.010	
Cast Iron, Gray	<=150 BHN	10, 90C	400-800	.005-.011	
Cast Iron, Nodular (ductile)	150-250 BHN (<=25 Rc)	10, 66C	300-500	.004-.010	
Cast Iron, Malleable	250-350 BHN (25-38Rc)	10, 66C	350-700	.004-.010	
Inconel	718	66C	150	.0010-.0015	
Stainless Steel	15-5	66C	200-300	.0025	
	17-4	66C	450	.003	
	200-300 Series	66C	250-500	.003-.005	
	304	66C	425-550	.0040-.0055	
	400 Series	66C	350-600	.003-.006	
	410	66C	340-600	.004-.006	
Steel	No/Low Alloy CRS, HRS	66C	800-900	.0015-.0035	"Pleated", or W-shaped chips can be catastrophic here. Reduce feedrate to create smooth chipping.
	Carbon, <=38 RC	66C 90C	400-800	.005-.009	
	Tool, Med Alloy, 12-38 Rc	66C 90C	250-750	.004-.010	
	High-Temp Alloy, 12-38 Rc	66C 90C	100-350	.003-.006	
Stellite L605		66C	150	.0025	
Titanium		66C	100-210	.0025-.0030	

**Unless noted, use the same grade insert in all pockets of the drill.*

Customer Service and Technical Support: *800.497.1892* ◆ ***Fax:*** *419.738.6517*
www.kyocera.com/cuttingtools ◆ *otmsales@kyocera.com*

DRILLING SPEEDS AND FEEDS
STINGER™ SMALL DIAMETER DRILLS

NOTES: 1. Do <u>not</u> overtighten the insert screws in these drills. Over-tightening the screws will break the inserts.
2. Do <u>not</u> drill stacked plates with these drills.

MACHINING RECOMMENDATIONS

Material	Condition	Insert Grade*	SFM	FPT
Copper		5N	900-2000	001-.003
		13	800-1900	.001-.003
High Temp Alloys		13	100-175	.0005-.0008
Inconel	600	13	250	.0008-.0012
	718	13	150-250	.0015-.0025
Stainless Steel	17-4	13	450	.0009-.0012
	300 Series	23C	300-500	.0012-.0015
		40C	400-600	.0012-.0015
	304	23C	300-600	.0008-.0012
		40C	400-700	.0008-.0012
	400 Series	23C	400-600	.0012-.0015
		40C	500-700	.0012-.0015
Steel, Low Carbon	Annealed	5N	600-900	.0012-.0018
		40C	500-800	.0012-.0018
	Carburized, 35-50 Rc	5N	300-400	.0012-.0018
		40C	250-350	.0012-.0018
Steel, Medium Carbon	17-32 Rc	5N	500-700	.0012-.003
		40C	400-600	.0012-.003
Titanium Alloys		13	150-250	.0008-.0012
Tool Steel	Annealed	5N	350-450	.001-.0015
		5N	150-250	.0007-.0012
4130, 4140, 4150	Annealed	5N	500-700	.0012-.003
	35-50 Rc	40C	400-600	.0012-.003
6150	Annealed	5N	500-600	.0012-.0015
		40C	400-600	.0012-.0015
Aluminum		5N	1000+	.002-.006
		13	900-1800	.002-.006
Brass		5N	900-1500	.001-.003
		13	800-1400	.001-.003
Bronze		5N	500-900	.001-.003
		13	400-600	.001-.003
Cast Iron	Gray	5N	500-900	.001-.003
		13	300-500	.001-.003
		23C	300-500	.001-.003
	Nodular (Ductile)	5N	500-700	.001-.003
		13	300-500	.001-.003
		23C	350-450	.001-.0022
	Malleable	5N	400-800	.001-.003
		13	300-500	.001-.003
		23C	300-500	.001-.003

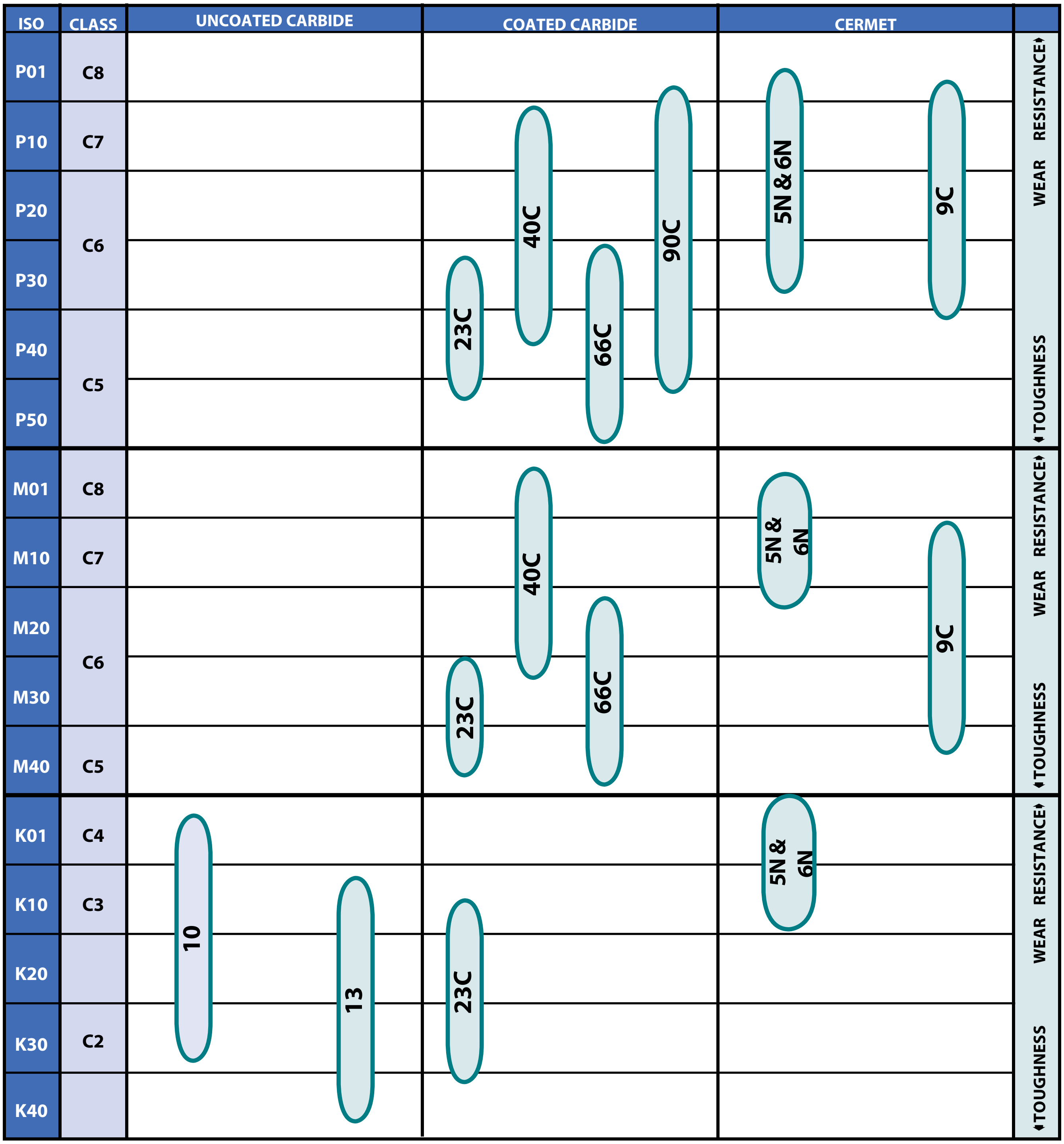
**Unless noted, use the same grade insert in all pockets of the drill.*

INSERT GRADES FOR DRILLING

GRADES		COMPOSITION & COATINGS	APPLICATION RECOMMENDATIONS
COATED CARBIDE GRADES	23C	TiN TiC TiN C5 Base Substrate	Tough CVD coated carbide grade for demanding applications in steel, stainless steel and steel castings. Takes shock and interruptions well.
COATED CARBIDE GRADES	40C	TiN Al_2O_3 TiCN C5 Base Substrate	CVD coated. Thick wear resistant coating layers over a cobalt enriched substrate provide high wear resistance with good edge security, drilling in most steels.
COATED CARBIDE GRADES	66C	TiN C5 Base Substrate	Very tough PVD coated grade for drilling alloy steel, stainless and tougher materials.
COATED CARBIDE GRADES	90C	TiN TiCN C6 Base Substrate	CVD coated grade for drilling applications in steel requiring toughness with good wear resistance.
UNCOATED CARBIDE	10	C2 Substrate	Uncoated carbide grade for drilling in gray cast iron and non-ferrous materials.
UNCOATED CARBIDE	13	C2 Substrate	Micrograin uncoated carbide for drilling heat resistant and titanium alloys.
CERMET GRADES	5N	Titanium carbonitride with metal binders	Tough cermet with good wear resistance for drilling in most steels. Not for center pocket. (Except for the Stinger Drills)
CERMET GRADES	6N	Titanium carbonitride with metal binders	Tough cermet with good wear resistance for drilling in most steels. Not for center pocket.
CERMET GRADES	9C	TiN Titanium carbonitride with metal binders	PVD coated cermet for roughing to finishing of steels. Not recommended for center pocket.

Customer Service and Technical Support: 800.497.1892 ◆ **Fax:** 419.738.6517
www.kyocera.com/cuttingtools ◆ otmsales@kyocera.com

CLASSIFICATION CHART FOR DRILLING GRADES



ISO DESIGNATIONS	
P	STEELS, TOOL STEELS
M	STAINLESS STEELS
K	CAST IRON & NON-FERROUS METALS

C CLASSIFICATIONS	
C-1 through C-4	C-5 through C-8
Abrasion resistant grades. Machining cast iron & short chip materials.	Crater resistant grades. Machining steel & continuous chip materials.
C-1 Roughing C-2 General Purpose C-3 Semi-Finishing C-4 Finishing & Precision Boring	C-5 Roughing & General Purpose C-6 Semi-Finishing C-7 Light Finishing C-8 Finishing & Precision Boring

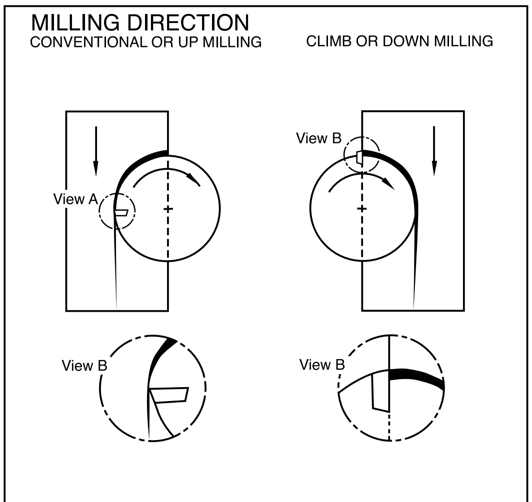
MILLING BASICS

BASIC MILLING

Most machining benefits from rigid set-ups. Milling applications benefit greatly. Short adapters and rigid fixturing will greatly increase tool life and feed capabilities.

CLIMB MILLING VS. CONVENTIONAL MILLING

Climb mill whenever possible. The inserts enter the cut at full feed per tooth, and exit as the chip thins to zero. Less heat is generated and work hardening is minimized. Today's inserts can easily take the forces generated, and free-cutting positive geometries won't shake the machine to pieces. Exceptions to this are manual machines and worn out CNC machines, where conventional milling is necessary because of backlash.



CLIMB MILLING IS GENERALLY RECOMMENDED OVER CONVENTIONAL MILLING. IN CLIMB MILLING, THE CUTTING FORCES TEND TO PUSH THE WORKPIECE AGAINST THE FIXTURE WHICH REDUCES THE POSSIBILITY OF VIBRATION.

POSITIVE VS. NEGATIVE GEOMETRY

Traditional indexable mills have used negative inserts extensively. Today's mills use positive inserts. Positive inserts have only half, or less, usable edges than negative inserts, but the free cutting action of positives usually outweighs the effects of edges lost. Today's insert compositions and edge geometries make strong, free-cutting edges that hold up in the cut. Feedrates are higher than negative inserts, horsepower requirements are lower, and tool life is greater. In the end, the cost per part produced is generally lower using positive inserts cutting most materials.

FACE MILLING

Whenever possible, face with a 30-degree lead mill. This will cut more freely, lowering horsepower requirements, and increasing tool life. The only time face milling with a 90-degree lead mill is appropriate is when a square corner is required along a cut.

PROFILE MILLING

Profile milling is a common milling application. The speeds below apply to profile milling. The feedrates can be increased, in some cases dramatically. The reason is that when the width of cut is less than half the cutter diameter, chip thinning occurs. Higher feed rates are actually required to create the necessary feed per tooth.

SLOTTING

Slotting is the most difficult milling application. The cutter begins cutting a thin chip, continues into a thick chip, then exits the cut through a thin chip again. The chip has one-half revolution to get free of the flute. Chip evacuation is critical, and chips can pile up behind the cut until they fall back into the mill and get re-cut. Re-cut chips will destroy insert edges quickly, leading to possible catastrophic tool failure. Deeper slots are even more difficult. Compressed air should be used to move all chips out of the work area, and feedrates must be appropriate for the material but not excessive.

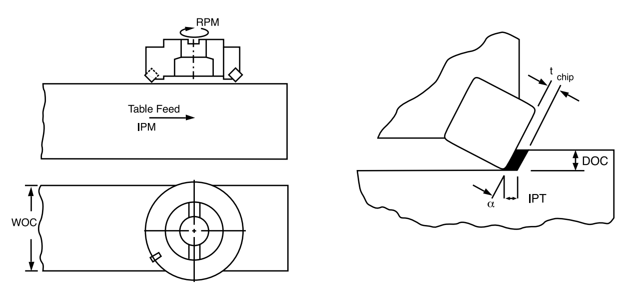
LEFT-HAND MILLS

OTM has a large selection of standard left-hand mills designed specifically to utilize the unused portions of the inserts from right-hand mills. Economy is the driving force of this design.

TIPS AND TRICKS

When milling around sharp corners, whenever possible interpolate the corner, using the corner as the center of the arc and the cutter radius as the arc radius. This will keep the cutter loaded through the corner, eliminate the exit and entrance cuts, and not leave a burr on the corner. Insert life will improve, and it's also a shorter distance to travel.

When making multiple passes to get to depth, don't divide the depth into equal increments. Instead, vary the DOC with each pass. This will increase insert life by reducing notch wear of the insert. (Notch wear occurs at the point the insert intersects the top edge of the cut.)



MILLING CHECKLIST

1. Properly identify the milling task process by reviewing part print parameters by dimension and material to be machined.
2. Identify the best machine tool and process for optimum performance.
3. Review the machine capabilities by:
    - a. Rigidity of set-up
    - b. Horsepower
    - c. Axis travel
    - d. Speed and feed limits
    - e. Overall condition
    - f. Spindle configuration
4. Select a cutter:
    - a. By style (facing, chamfering, etc.)
    - b. By task:
        - 1) Cut diameter
        - 2) Lead Angle
        - 3) Mounting method
        - 4) Hand
    - c. Select inserts by proper grade, geometry and top prep.
    - d. Select machining data:
        - 1) Recommended surface speed (ft/min)
        - 2) Calculate RPM
        - 3) Select feed rate per tooth (when radial depth of cut is less than 50% of the diameter, chip load should be increased accordingly)
        - 4) Check K factor (effective number of teeth in cut)
        - 5) Calculate table feed
        - 6) Calculate metal removal rate and power consumption

OTHER CONSIDERATIONS

Coolants

Coolants are **not recommended for milling with cermet and ceramic inserts.** Coolant exposes the inserts to damage from thermal shock caused by the intermittent heating and cooling cycles. Directed air pressure or sometimes an aerated mist can aid in chip flow.

Cutter Handling

A cutter should be maintained in good condition, making sure the cutter is properly cleaned and changing spare parts.

Insert Care, Mounting and Indexing

With the exception of single point cutters, each milling insert is dependent on the others in the cutter. Failure of one insert increases chip load on the others, usually resulting in severe breakage of the rest.

To insure proper chip load on all inserts, care must be taken to seat them properly into the pocket.

When mounting new inserts or indexing, the pockets and/or cartridges must be thoroughly cleaned of any loose material.

Milling inserts must be eventually replaced due to failure by wear or fracturing. Observation of insert condition during use, and the nature of it, will indicate satisfactory or unsatisfactory performance.

MOUNTING CONSIDERATIONS

Integral shank cutters offer the best rigidity. Use appropriate style end mill holder or collet chuck with non-pull collets. Special continuous clamping milling chucks usually result in the most rigid set-up. On shell mills, make sure arbor is proper size, type and in good repair. Run-out, chatter, poor surface finish and poor tool life can sometimes be traced to poor or improper mounting.

MILLING METHOD CONSIDERATIONS

Down milling (climb milling), or milling in the direction of the feed, should always be the first choice. In down milling, the insert enters through the un-milled surface, and produces a chip that becomes progressively thinner as the insert pushes through the part, which helps in materials that tend to work-harden. Down-milling also places downward pressure on the workpiece and forces the work in the direction of the feed. The thin chip exit helps insert edge security, better tool life and of considerable importance in milling heat resistant materials.

FEED / SPEED CALCULATIONS

UNIT CONVERSIONS

Feed and speed calculations use the following abbreviations:

DEFINITION OF TERMS

WOC = WIDTH OF CUT (INCHES)
DIA = CUTTER DIAMETER (INCHES)
DOC = DEPTH OF CUT (INCHES)
EFF = MACHINE EFF
f = FEEDRATE (SEE IPM, IPR, AND IPT)
HPM = HORSEPOWER REQUIRED AT THE MOTOR (HP)
HPS = HORSEPOWER REQUIRED AT THE SPINDLE (HP)
IPM = FEEDRATE (INCHES PER MINUTE)
IPR = FEEDRATE (INCHES PER REVOLUTION)
IPT = FEEDRATE (INCHES PER TOOTH)
LOC = LENGTH OF CUT (INCHES)
N = NUMBER OF EFFECTIVE TEETH IN CUTTER
Q = METAL REMOVAL RATE (CUBIC INCHES PER MINUTE)
RPM = REVOLUTIONS PER MINUTE
SFM = SURFACE FEET PER MINUTE
T = TIME (SECONDS)
t_{chip} = CHIP THICKNESS (INCHES)
UHP = UNIT HORSEPOWER FACTOR
α = LEAD ANGLE

Cutting cycle time=length of cut (inches)/IPM
Cubic inches removed per minute =IPM x WOC x DOC

SURFACE SPEED

SFM = .262 x DIA x RPM

REVOLUTIONS PER MINUTE

$$RPM = \frac{3.82 \times SFM}{DIA}$$

FEEDRATE (INCHES/REVOLUTION)

IPM = IPT x N x RPM

FEEDRATE (INCHES/REVOLUTION)

$$IPR = \frac{IPM}{RPM}$$

FEEDRATE (INCHES/TOOTH)

$$IPT = \frac{t_{chip}}{\cos \alpha}$$

METAL REMOVAL RATE

Q = WOC x DOC x IPM (in³/min)

TIME IN CUT (IN SECONDS)

$$T = \frac{15.7 \times DIA \times LOC}{SFM \times IPT \times N}$$

or

$$T = \frac{60 \times LOC}{IPM}$$

OPTIMUM FEED RATES

The following formula can be used to determine optimum feed rates when chip thinning occurs:

$$IPM = \frac{\text{desired FTP}}{\dfrac{\sqrt{(\text{Cutter DIA} - \text{WOC}) \times \text{WOC}}}{\text{Cutter RADIUS}}} \times \text{effective teeth} \times \text{RPM}$$

The **lead angle** of the cutter also affects chip thinning. Only when using a 90-degree lead mill is the chip thickness equal to the feed advance per tooth. When the lead angle is 30 degrees, the chipload is actually only 87% of the feed advancement, resulting once again in less than optimum metal removal rates. Using the cosine of the lead angle the correct FPT can be determined.

FPT = desired FPT / COS (lead angle)

These two formulas appear to be a lot of trouble over relatively small gains, but when proper feedrates are applied, tool life will increase, cycle time will drop, and better part quality will result.

CHIP THINNING

In milling, the feedrate is also affected by the width of cut (WOC). As long as the WOC equals or exceeds the radius of the cutter, the FPT will equal the chip thickness. Whenever the WOC is less than the radius of the cutter, chip thinning occurs, meaning the chip thickness is less than the FPT (feed advance per tooth).

This results in lower metal removal rates, less efficient machining, higher cutting temperatures, and early insert failure.

MILLING FEEDS AND SPEEDS

MACHINING RECOMMENDATIONS

Material	Condition	Insert Grade	SFM	FPT	Comments
Aluminum	Low Silicon (< 8%)	5, 6, 13	1500-1800	.005-.007	
		12	2000-4/5000 SFM or 10K RPM or so max	.005-.020	Use Coolant
Aluminum, Aluminum-Bronze	High Silicon (> 8%)	66C	800-1100	.005-.008	
Bronze		1, 5, 6	1000-1100	.005-.008	
Bronze	Malleable	1, 5, 6, 10	400-700	.003-.006	
	Gray	10, 23C, 93C	600-1200	.004-.016	
	Nodular (Ductile)	1, 5, 6, 10	600-850	.004-.008	
Copper		1, 5, 6	1200	.007	
CPM 9V	9% Vanadium, 5.25% Chromium	23C	310-400	.005-.006	Flood Coolant
D2	Annealed	1, 5, 6	400	.002	
Inconel		93C	100	.003	0.060 DOC Max
	718	10	100-200	.001-.002	0.100 DOC Max
Invar 36	30% Nickel	93C	100-250	.003-.004	
Stainless Steel	Ferritic Martensitic	5, 6, 23C, 66C, 93C	500-800	.002-.006	
	Cast	5, 6, 23C, 66C, 93C	500-800	.003-.006	
	304	5, 6, 23C, 66C, 93C	400	.003	
	316L	5, 6, 23C, 66C, 93C	300	.003-.004	
Steel, Unalloyed		1, 5, 6	800-1400	.003-.006	
Steel, Low Alloy	Annealed	1, 5, 6	600-1200	.003-.006	
	Hardened	1, 5, 6	400-700	.002-.005	
Steel, High Alloy	Annealed	1, 5, 6	400-700	.003-.006	
	Hardened Rc	1, 5, 6	250	.002	
Steel, Cast	Low Alloy	1, 5, 6	600-1200	.004-.008	
	High Alloy	1, 5, 6	400-700	.003-.006	
Titanium		93C, 23C, 13	100-210	.002-.003	
Tungsten		6	400	.002-.004	

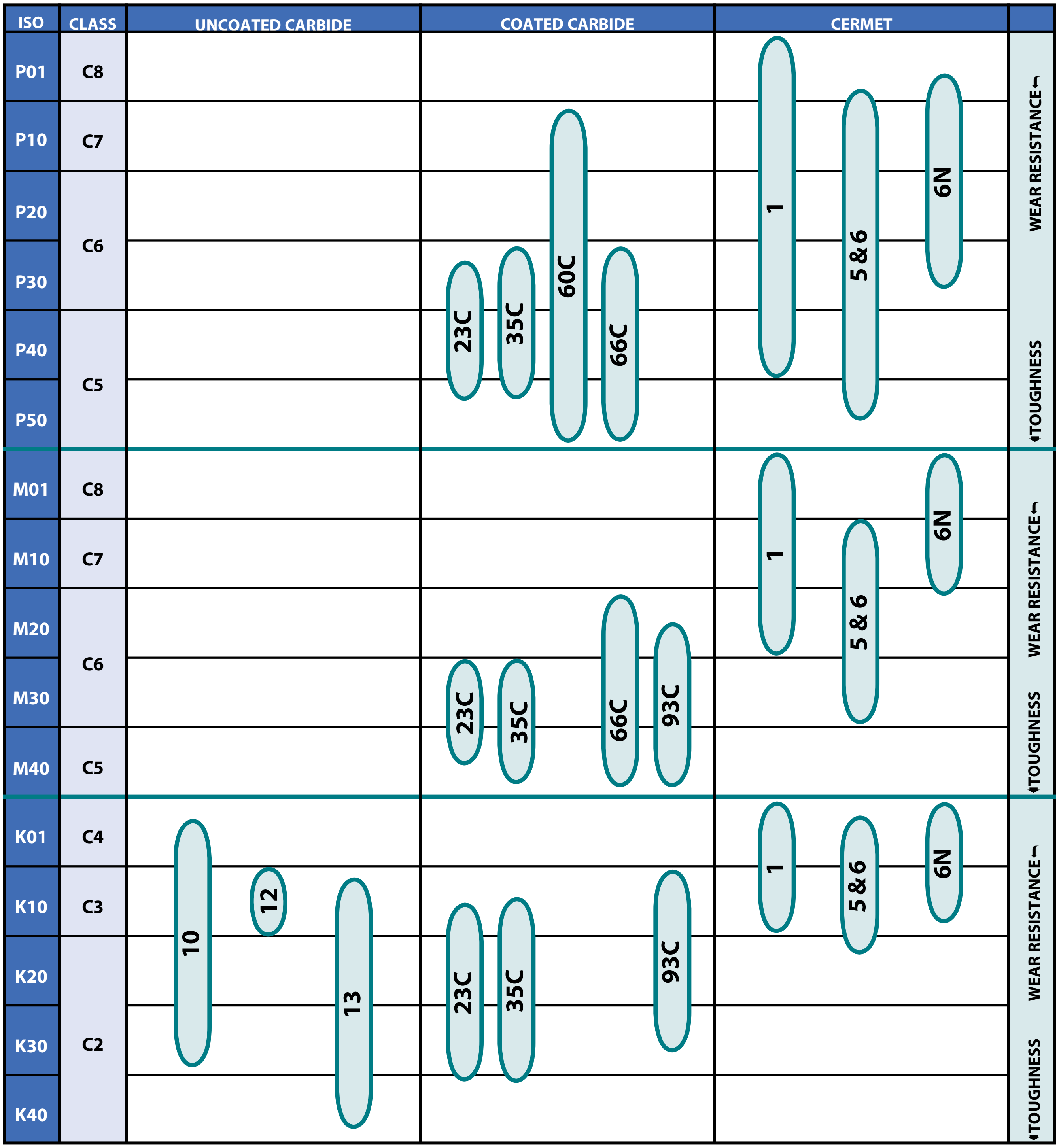
NOTE: *Do not use coolant on Grades 1, 5, or 6 unless specfically noted.*

LONG EDGE MILLING

Slot milling with long edge mills is not recommended. When using long edge mills to circular interpolate inside diameters, allow sufficient room for chip clearance. Use of forced, directed air may be required in all cases. Depth of cut must be carefully considered relative to the machine horsepower.

Speeds and feeds shown above are suggested starting points. Actual speed and feeds may vary due to material, machine and operating conditions. Test cuts are recommended, checking machine and holding rigidity prior to cutting at the high range of recommended speed and feed. As depth of cut increases and/or full slotting, modify speed and feed downward accordingly. All speeds and feeds that apply to cermets should be reduced 10-20% when using carbide grades.

CLASSIFICATION CHART FOR MILLING GRADES



ISO DESIGNATIONS	
P	STEELS, TOOL STEELS
M	STAINLESS STEELS
K	CAST IRON & NON-FERROUS METALS

C CLASSIFICATIONS	
C-1 through C-4	C-5 through C-8
Abrasion resistant grades. Machining cast iron & short chip materials.	Crater resistant grades. Machining steel & continuous chip materials.
C-1 Roughing C-2 General Purpose C-3 Semi-Finishing C-4 Finishing & Precision Boring	C-5 Roughing & General Purpose C-6 Semi-Finishing C-7 Light Finishing C-8 Finishing & Precision Boring

Customer Service and Technical Support: *800.497.1892* ◆ ***Fax:*** *419.738.6517*
www.kyocera.com/cuttingtools ◆ *otmsales@kyocera.com*

INSERT GRADES FOR MILLING

	GRADES	COMPOSITION & COATINGS	APPLICATION RECOMMENDATIONS
COATED CARBIDE GRADES	23C	TiN TiC TiN C5 Base Substrate	
	35C	TiN Al_2O_3 TiC TiN C5 Base Substrate	CVD coatedgrade for milling steel, stainless steel and cast steel.
	60C	TiN Al_2O_3 TiCN C5 Base Substrate	Tough CVD coated grade for roughing and semi-roughing at low to medium speeds in steel.
	66C	TiN C5 Base Substrate	PVD coated grade as a first choice in milling tough to machine stainless steels and other tough materials.
	93C	TiCN C6 Base Substrate	PVD coated micrograin carbide for sharp edge applications when wear resistance is required. Recommended for tough stainless steel, high temp and titanium alloys.
UNCOATED CARBIDE	10	C2 Substrate	Micrograin uncoated carbide recommended as first choice in milling gray cast iron. Works well in most aluminum and non-ferrous materials, and as an alternate choice for high nickel/high temp alloy materials.
	12	C3 Substrate	Micrograin uncoated carbide primarily for milling aluminum.
	13	C2 Substrate	Micrograin uncoated carbide recommended first heat resistant and titanium alloys; also aluminum and non-ferrous materials.
CERMET GRADES	1	Titanium carbonitride with metal binders	A newly developed “super cermet” grade for rough milling of steel, tool steel and some stainless steels. Secondary application in cast iron, aluminum and other non-ferrous materials. Has toughness and shock resistance paralleling Grade 6, but with better wear resistance.
	5	Titanium carbonitride with metal binders	Tough cermet, but also exhibits good wear resistance. Recommended primarily for steel, tool steels and some stainless steels. Also shows some versatility in a variety of other materials.
	6	Titanium carbonitride with metal binders	Toughest cermet with good shock resistance for rough milling in a wide range of materials. Primarily recommended for tool steel applications
	6N	Titanium carbonitride with metal binders	Toughness combined with good wear resistance. Covers a wide range of applications and handles interruptions well.